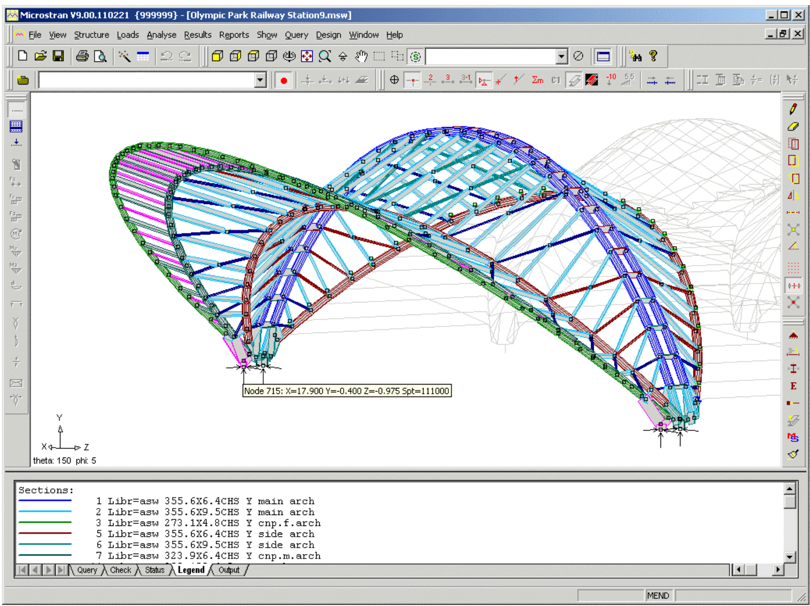
Microstran V9

User Manual

Engineering Systems



DISCLAIMER
Subject to limitations imposed by law, Engineering Systems makes no warranty of any kind in connection with the licensed material. Engineering Systems shall not be liable for any errors contained in the licensed material nor for any incidental or consequential damages resulting from the use of the licensed material. Engineering Systems is not engaging in the provision of consulting services in supplying the licensed material. Users of the licensed material are advised that output from computer software should be subjected to independent checks. Engineering Systems reserves the right to revise and otherwise change the licensed material from time to time without notification, or provision of revised material.

SOFTWARE LICENCE
The software is supplied to the user under licence. It may be installed on as many computers as required but the number of concurrent users must not exceed the number of licences held. For network licences, use is permitted only in the country for which the licence was supplied. The software may not be sub-licensed, rented, or leased to another party. The licence can only be transferred to another party on payment of a transfer charge determined by Engineering Systems.

Engineering Systems Pty Ltd
14 Eastern Road
PO Box 85
Turramurra NSW 2074
Australia

Tel: +612 9488 9622
Fax: +612 9488 7883
E-mail: support@microstran.com.au
Web: www.microstran.com.au

July, 2012

Olympic Park Station
The front cover features the multi-award winning railway station built for the Sydney 2000 Olympic Games.
Architect: Hassell
Engineer: Tierney & Partners
Microstran model provided by courtesy of Tierney & Partners. Photography by Claire Morgan.

Preface

Microstran is a comprehensive software package for the analysis and design of all kinds of frame structures – beams, trusses, frames, towers, and bridges. The structures you can analyse with Microstran may be two-dimensional or three-dimensional, varying in size from a few members to many thousands of members.

Chapter 1 – "Introduction" provides an overview of the capabilities of Microstran. Whether you are installing Microstran for the first time or updating an existing system, you will find all the necessary information in Chapter 2 – "Getting Started". Chapter 12 – "Input Tutorials" contains a number of exercises for guiding the new user through most of the basic operations required for doing a job. These tutorials emphasize the use of graphical input techniques that have become very important in modern structural software. Some of the tutorials are available on the Microstran website as "self-playing" demos. Chapter 3 – "Menus and Toolbars" provides a summary of the commands available and other chapters provide reference and technical information.

This manual is available to the Microstran user on-line, together with comprehensive "pop-up" help for dialog boxes. The on-line help system provides a synchronized table of contents and powerful methods of searching for topics.

If the file Readme.txt is present in the Microstran folder after installation, you should read it for information that became available after the manual was printed. The file is automatically displayed during installation but it may be displayed in Notepad at any time by double-clicking the file in Windows Explorer.

The first version of Microstran fully compatible with this edition is V9.01.

Contents

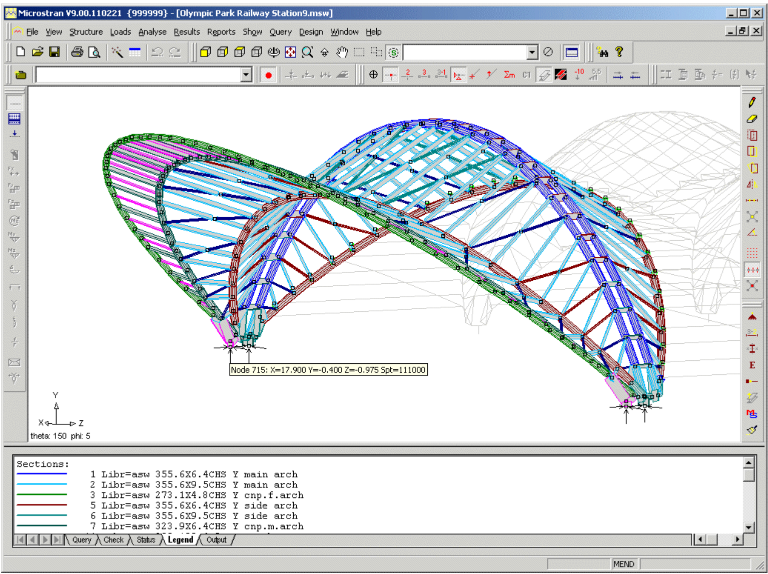
8:Table Input 151

9:Archive File Input 167

1:Introduction

Microstran is a comprehensive software package for the analysis and design of 2-D and 3-D frames and trusses. This chapter gives an overview of the principal features of the system while more details are provided in other chapters.

The Microstran Window



THE MICROSTRAN WINDOW

Main and Output Windows

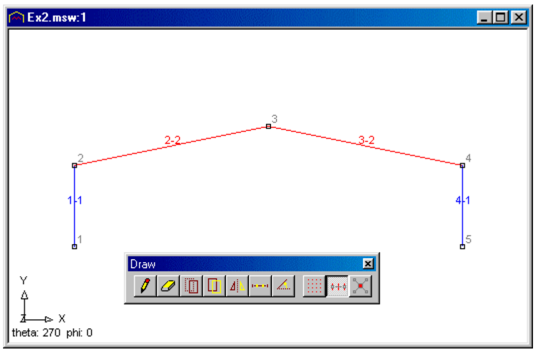
Microstran has a *main graphical view* bordered by a menu and toolbars at the top, toolbars on both sides, and an output window at the bottom. The output window contains tabs for display of *Query* results, *Check* output, *Status* information, *Legends*, and *Output* from ancillary programs. This gives rapid access to all the information about the current job. The menu bar, toolbars, and output window are all “dockable”. Each may be dragged to another edge of the main window, or floated inside or outside the main window.

The Menu Bar

The menu bar, normally at the top of the Microstran screen, is the principal command interface. Each menu title gives access to a drop-down menu of commands. In general, each toolbar button corresponds to a menu item. See Chapter 3 – "Menus & Toolbars" for a description of each of the drop-down menus.

Toolbars

In addition to the main toolbar in the top left corner, Microstran has View, Display, Draw, Attributes, Load Input, Results, and Help toolbars. The Steel Design, OK, and Extra toolbars may be displayed with the **View > Toolbars** command. The accompanying diagram shows the Draw toolbar positioned conveniently for graphics input.



FLOATING TOOLBAR

Microstran toolbars have a number of additional features:

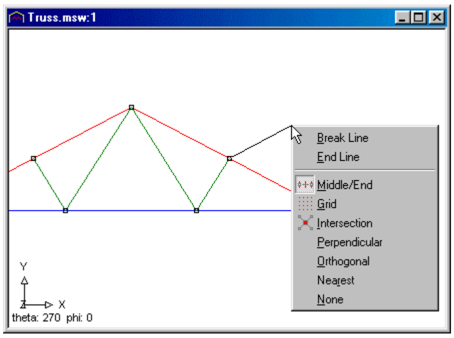
- Tooltips – a pop-up window shows help for each button when the cursor rests over the button.
- They are customizable – buttons may be added, deleted, or moved.
- Optional large buttons – these are useful at higher screen resolutions.

See Chapter 3 – "Menus & Toolbars" for more information.

Context Menus

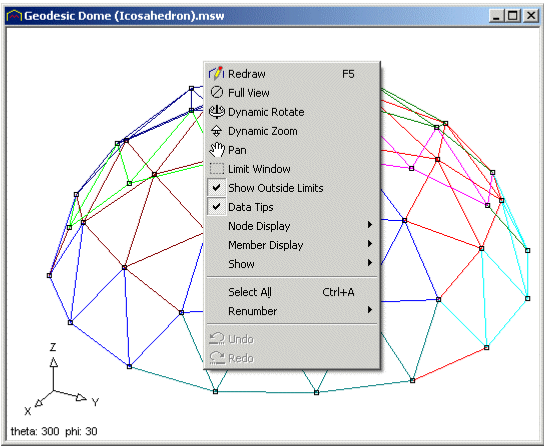
A context (or pop-up) menu appears when you click the right mouse button, giving rapid access to relevant menu commands without the need to search through the drop-down menus. Context menus are fully implemented throughout Microstran.

The diagram below shows the context menu that appears when you right-click during a Draw operation. This menu allows you to break the line (so you can start somewhere else), end the line (stop drawing), or change the snap mode.



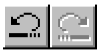
CONTEXT MENU FOR DRAW COMMAND

The diagram below shows the context menu that appears when you right-click in the main window with the cursor not close to a node or a member. This context menu provides a convenient shortcut to commonly used commands for viewing and manipulating the structure data.



MAIN CONTEXT MENU

User Interface Features

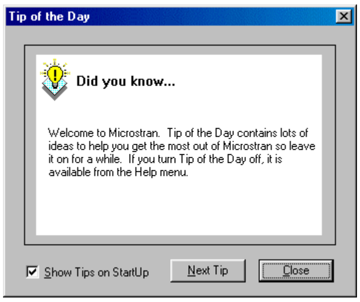


Undo / Redo

Undo makes your work easier. As well as being able to recover from mistakes, you can experiment with unfamiliar operations, knowing that it only takes one click to get back to where you were. There is only one undo level.

Tip of the Day

The Tip of the Day dialog box appears when you start Microstran.



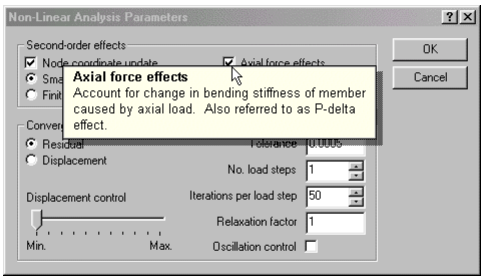
TIP OF THE DAY

Context-Sensitive (Pop-Up) Help

Context-sensitive Help gives quick access to information about dialog box items simply by clicking the question-mark button in the title bar and then clicking the item. Many dialog boxes in Microstran have context-sensitive help, a typical example of which is shown below.

The Windows component necessary for pop-up help, WinHlp32.exe, is not included in Windows Vista and Windows 7. It may be downloaded from the Microsoft website, see: http://support.microsoft.com/kb/917607

Policy defaults on a domain based network may block the use of .hlp files over a network. The administrator can modify domain policy to permit this if required.



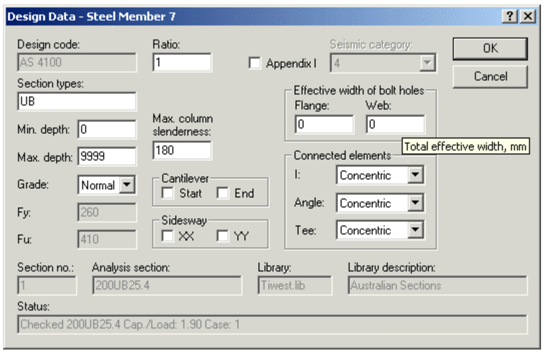
CONTEXT-SENSITIVE (POP-UP) HELP

Notes

When you work on a Microstran model over a period of time it is helpful to have a few notes explaining any changes you make. The **Structure > Notes** command allows you to store easily accessible descriptive information with the job.

Tooltips in Dialog Boxes

In addition to the pop-up help available through the Help button, dialog boxes have tooltips. The tooltips in dialog boxes are similar to those that appear when the mouse cursor passes over a toolbar button.



TOOLTIP IN A DIALOG BOX

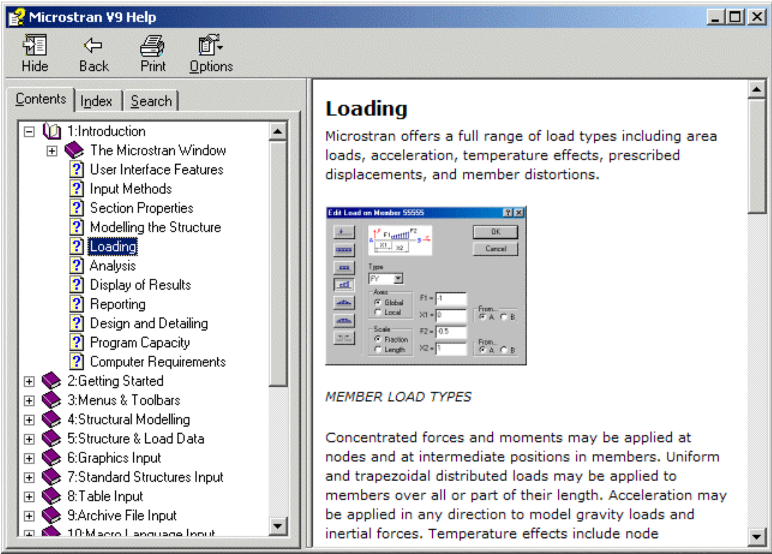
Data Tips

When the cursor rests over a node, a pop-up window shows the node number, the coordinates, and support conditions (see screen shot on page 1). For members, the data tip shows the member number, the node numbers, section and material numbers, and releases.

On-Line (HTML) Help

Limcon's on-line help allows you to browse help topics, look through an index, or do a full-text search for any word or phrase. The entire Microstran User Manual is available through on-line help.

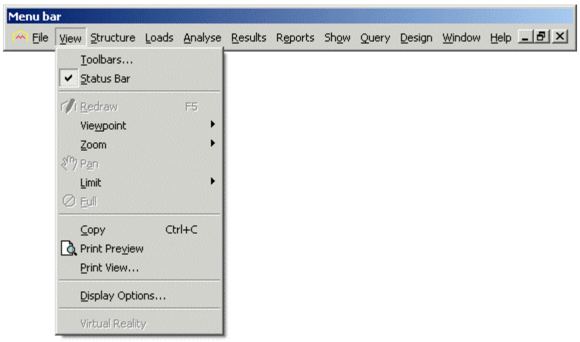
Windows security prevents HTML help files (.chm files) being accessible over a network. This means that if the Microstran \Program folder is on the network rather than a local disk HTML help will only be available if the Mswin9.chm file is on a local disk.



ON-LINE HELP

Icons on Menus

Microstran displays a small icon next to each menu item that has a toolbar button equivalent. This aids recognition and provides a visual cue to help you relate the toolbar command and the menu command.



ICONS ON MENUS

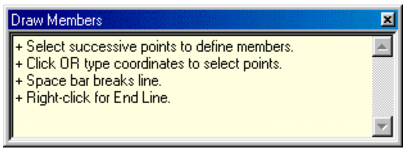
Print Preview

Print previewing is available for graphics and reports. This allows you to make sure the output is what you want before printing it. The preview shows exactly what you will get on the printed page so you can check margins and page orientation without wasting paper.

Smart Help – The Command Assistant

First-time users need intelligent help. The Command Assistant is a pop-up help window that appears when you begin one of several common graphical tasks, such as drawing or copying members. The Command

Assistant window appears automatically just when the first-time user is wondering what to do next. It disappears at the end of the command. You can turn off this feature when you no longer require assistance. Here is the Command Assistant window that appears during the Draw command.

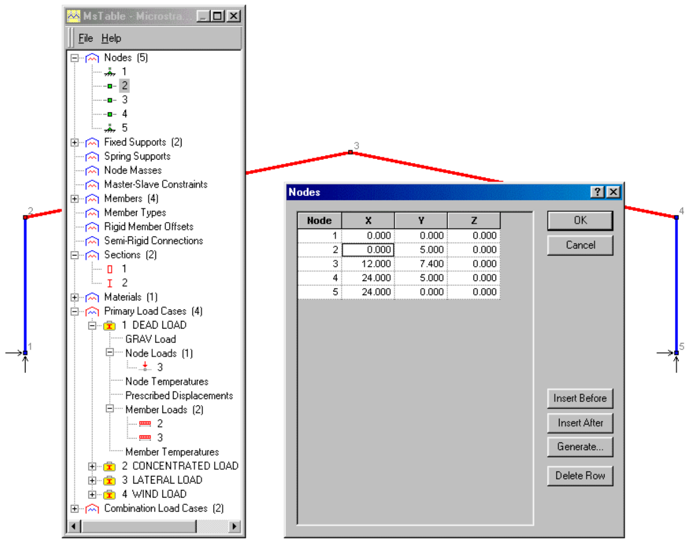


THE COMMAND ASSISTANT

Table Input

When you select Table Input all job data is displayed in a *tree view* similar to that in Windows Explorer. The tree view provides a very convenient overview of all the data for the current job. The icon next to each item in the tree view is a visual cue to its type. Note, for example, how section types and load types are clearly identified in the diagram below. Double-clicking any data item in the tree displays a table of all such items so you can easily edit them.

There is a data table for every type of data entity. Data may be transferred from or to the tables using Copy (Ctrl+C) and Paste (Ctrl+V). Undo (Ctrl+Z) may also be used. The rows of each table may be sorted into ascending order by double-clicking on the header of the column by which the rows are to be sorted. Double-clicking again reverses the sort order.



TREE VIEW AND DATA TABLE

Shortcut Keys

Shortcut keys, also known as *accelerator keys* give you the option of using a single keystroke for some commands. See “Shortcut Keys” on page 23.

Enhanced Metafile Format (EMF) on Clipboard

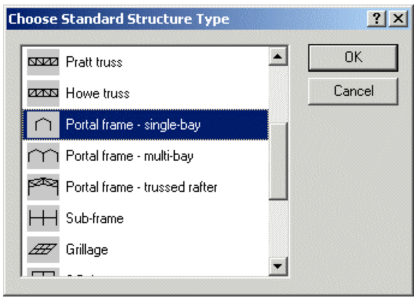
Microstran facilitates transfer of images to CAD and other programs by using the Enhanced Metafile Format (EMF) for the Windows clipboard. When you select the **View > Copy** command Microstran stores the screen image on the clipboard in EMF format. In programs such as AutoCAD, you can use the **Paste** command to directly insert this image. Alt+PrtScr writes a Windows *bitmap* to the clipboard. Both of these formats may be *pasted* into Microsoft Word documents.

Input Methods

Many different input methods are available to describe the structure and its loading. The different methods may be used interchangeably so that it is possible to use the method that is most convenient for each particular input task. For example, tabular node and member data may be input using Table Input and then Graphics Input may be used to define section numbers, selecting members visually.

Graphics Input allows you to specify structure and load data graphically. Using the mouse, members may be drawn, erased, moved, copied, or sub-divided. Graphics Input operates essentially as an intelligent CAD system specially designed for the task of structure and load definition. Powerful features are available for the generation of regular structures and loads are easily applied to groups of members or to a single member. Graphics Input is often used in conjunction with other input methods because of the ease with which nodes and members can be selected visually.

Standard Structures Input is often the easiest way to specify the input data for a range of common structure types. The structure type is selected and a small number of parameters that define the structure are then input. If your structure is similar to any of the standard types you may start with this method and then make any necessary changes with another input method, Graphics Input for example. The Standard Structures dialog box is shown below.



STANDARD STRUCTURES

Structure types available include:

- Continuous beams.
- Beam on elastic foundation.
- Trusses (triangular, parallel-chord).
- Portal frames (single-bay, multi-bay).
- Trussed-rafter portal frames.
- Sub-frames.
- Grillages (including grillage on elastic foundation).
- 2-D frames.
- 3-D frames.
- Trestles (K-bracing, X-bracing).
- Geodesic domes (icosahedron, octahedron, and tetrahedron).

Table Input allows you to enter structure and load data in tabular form. A tree control displays every item of job data in a convenient Explorer-like view. Double-clicking an item in the tree presents data in a table for editing. Buttons are provided to insert and delete rows and to generate multiple rows. With Table Input data may be exchanged between Microstran and other applications using the Windows clipboard. For example, node coordinates generated in a spreadsheet program, such as Microsoft Excel, may be pasted directly into the node table.

Archive File Input allows input to a text file called the *archive file*. The archive file format is used to save input as a single file for storage but it also provides a convenient format for editing. Other programs may create archive files to be imported into Microstran. The *automatic backup* facility writes a file in this format.

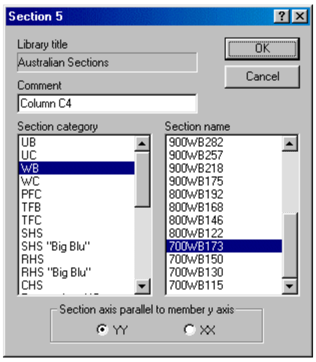
CAD Input allows importing structure data directly from any 3-D CAD system that uses the standard format AutoCAD DXF (Release 12). Structure data may also be exported from Microstran to a CAD system.

Macro Language Input operates from a data file called the *MLI file* that is prepared using a text editor. The built-in capabilities of the macro language for data generation and repetition allow MLI files to be very

compact. Input files used by some other programs (STAAD Pro, for example) are similar to MLI files.

Section Properties

Geometric section properties may be extracted from library files of standard steel sections, specified by section shape, or entered explicitly.



SELECTION OF LIBRARY SECTION

Microstran is supplied with a library of standard steel sections. A number of libraries are available for sections originating in countries including Australia, Japan, New Zealand, UK, and the US. For section shape input the required shape is selected from a range of common shapes and properties are computed automatically from the dimensions.

You may change any library or create a new one using the Section Library Manager.

Modelling the Structure

Gaps may be included in all numbering sequences – nodes, members, sections and load cases – with no operational penalty. Such gaps often simplify the data generation during input, facilitate the management of the data in the model for design and reporting, and more readily accommodate changes during the design process.

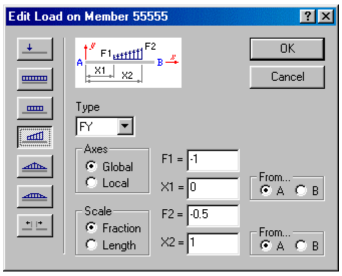
Note: In Microstran all entities have a number, referred to as a *label*. Labels may be any integer value from 1 to 99000.

Members may be modified in a number of ways to model different types of structural element. *Rigid member offsets* permit the accurate modelling of stiff joint regions and stiff elements such as deep beams and shear walls. The use of *semi-rigid connections* allows modelling of

joints that are neither pinned nor fully rigid. *Master-slave constraints* (kinematic constraints between nodes) facilitate the modelling of many kinds of structure such as “scissor” structures and floor slabs assumed to be rigid in-plane. *Tension-only members* and *compression-only members* may be used with non-linear analysis to model slender struts, such as “X”-bracing, and situations where lift-off occurs. The catenary cable option allows the correct modelling of structures containing *cable members*. The option for *gap and fuse elements* permits the modelling of gaps and members that yield or fail at specified load levels.

Loading

Microstran offers a full range of load types including area loads, acceleration, temperature effects, prescribed displacements, and member distortions.



MEMBER LOAD TYPES

Concentrated forces and moments may be applied at nodes and at intermediate positions in members. Uniform and trapezoidal distributed loads may be applied to members over all or part of their length. Acceleration may be applied in any direction to model gravity loads and inertial forces. Temperature effects include node temperatures, member temperatures, and temperature gradients. Member distortions may be used to model “lack of fit” or prestress and also to generate influence lines.

Cable members may be loaded with a restricted range of load types, which includes acceleration loads, uniformly distributed loads in global axis directions, member centroidal temperatures, and axial member distortions.

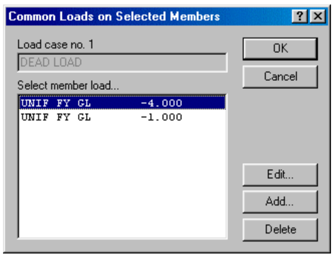
Combination load cases may be formed by factoring and combining primary load cases or other combination load cases. There is no limit either to the number of loads that may be applied to any member, or to the number of load cases.

The Moving Load Generator option will create the multiple load cases necessary to model a series of loads moving along a continuous beam or an area modelled as a grillage (e.g. a bridge deck).

Multiple Selection Load Editing

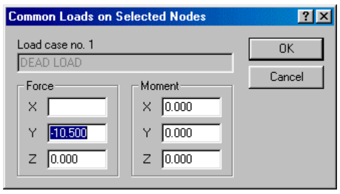
You may select several members when using the **Edit Member Loads** command. Member loads that are common to all the selected members are displayed in the list box. These may then be selected, one at a time, for editing or deletion. Changes to member loads in this dialog box are applied to all the selected members.

Note: Only loads present on ALL selected members are available for editing.



EDITING A LOAD COMMON TO MORE THAN ONE MEMBER

Multiple nodes may also be selected for the **Edit Node Loads** command. Where a numerical value is shown for a node load component in the node load input dialog box, that load is present on all selected nodes. A blank edit box indicates a node load component that is not the same on all selected nodes. Entering a value in a blank box and clicking OK will set the corresponding node load component for all selected nodes. The load components corresponding to any blank edit box will remain unchanged when you click OK.



EDITING A LOAD COMMON TO MORE THAN ONE NODE

Analysis

Microstran offers a number of types of analysis. These are *linear elastic analysis*, *non-linear elastic analysis*, *elastic critical load analysis*, and *dynamic analysis* with *response spectrum analysis*. All analysis modules use a highly efficient equation solver giving unsurpassed speed and efficient use of disk space. A *profile optimizer* ensures the maximum efficiency in solution of large and complex structures.

Non-linear analysis may be configured to take into account the non-linear effects of load displacement (P-Δ effect) and axial force in members (P-δ effect). The analysis may also be configured to use finite displacement theory instead of the conventional small displacement theory.

Elastic critical load analysis computes the elastic critical load for the structure and the true effective length for all members for selected load cases. Because the elastic critical load is dependent on the arrangement of load on the structure, these parameters will vary from load case to load case.

Dynamic analysis computes frequencies and the associated mode shapes for the natural vibration modes of the structure. Structure mass is computed automatically and additional mass may be specified as required. *Response spectrum analysis* converts base shears to stress resultants throughout the structure and is most commonly used to model earthquake loadings.

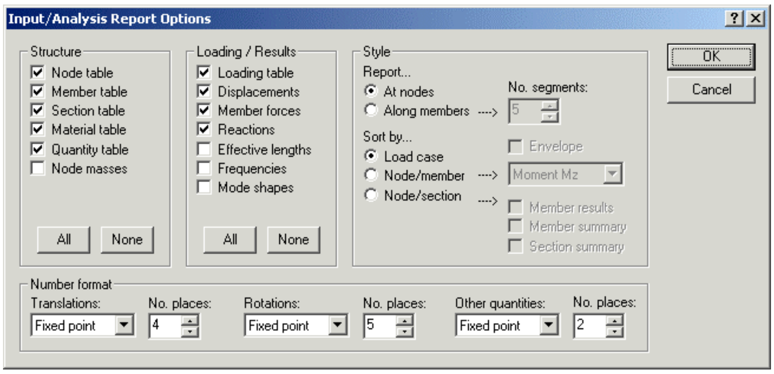
Display of Results

Microstran displays applied loads and analysis results graphically, with the option of annotating load and force diagrams with numerical values. Bending moment and shear force diagrams may be shown for either or both principal axes. Once the view shows the results required, you may print it by clicking a toolbar button. The **Preview** command shows exactly how the view will be printed.

Analysis results may also be displayed numerically, for any selected member, in the output window.

Reporting

Microstran includes a flexible report generator, which allows you to restrict the contents of analysis reports to just the information required. Results may be sorted by node and member or by load case, and values may be reported at nodes or intermediate positions along members. Envelopes of results for specified load cases may also be generated.



INPUT/ANALYSIS REPORT OPTIONS

Summaries available when you select “Sort by node/section” in the report options dialog box, above. With this setting node results are sorted by load case within node number and member results are sorted by load case within *section* number. You may independently include the member results in three different formats:
1. Full results for the members.
2. A summary (envelope) sorted by load case within member number.
3. A summary (envelope) sorted by load case within section number.

See “Input/Analysis Report Options” on page 267.

Design and Detailing

Integrated design options use Microstran analysis results to check or design members of the structure. Design modules are available for a range of national codes including:

Steel

- AISC ASD – 9th Edition (1989)
- AS 1250 / AS 3990
- AS 4100 – 1998
- AS/NZS 4600:2005
- BS 5950:2000
- NZS 3404 – 1997
- SSCJ/AIJ

Reinforced Concrete

- AS 3600:1994
- BS 8110

There are options for exporting information for steel detailing packages such as Xsteel.

The reinforced concrete detailing option generates a CAD DXF for beams and columns. This may be imported into a CAD program to make a detail drawing, which includes bar schedules.

Program Capacity

The capacity of Microstran depends on many factors including the type of structure, the number of load cases, and the memory and disk capacity of the computer. In appropriate circumstances, the capacity of the standard version is 35,000 members.

Computer Requirements

Recommended

- Windows XP or later.
- 1024 x 768 resolution or higher.
- 32-bit colour.
- Colour printer.

Virtual Reality Graphics

The Virtual Reality View button in Microstran displays a virtual reality representation of the structure. Virtual reality is only available on displays with 65535 or more colours. To see the colour capability of your display, go to the Settings page of Display Properties in the Windows Control Panel. The necessary minimum colour depth is High Colour (16 bit). Display adapter memory and screen resolution are the main factors affecting the colour depth available. If your display has the necessary colour depth and virtual reality graphics is still not available, you may need to set the OpenGL pixel format – see “Configuration” on page 20.

Note

VRML graphics requires Internet Explorer, or an equivalent browser, with a suitable VRML “plug-in” installed.

What's New in V9?

Input

- Merge jobs.
- Load case template.
- Copy member load function.
- Non-vertical (global axis) area loading.
- One-way area loads.
- Multiple load case delete.
- Renumber load cases.
- Extend Member command.
- Graphics input shortcut keys.
- Sets preserved when renumbering nodes or members.
- Conversion of job units.
- Enhanced job notes.
- New button for View > Limit > Set command.
- Section builder.

Graphics

- Faster results plotting.
- New button to plot reactions.
- Enhanced zoom and pan in main view and OpenGL view.
- Print OpenGL view, save .JPG image.
- Plotting connection symbols in main view.
- Improved handling of thick line plotting.
- Enhanced member data tips.
- Moving loads graphics.

Analysis

- Additional parameters for dynamic analysis.

Design

- Support for AS/NZS 4600:2005.
- Steel connection design to AS 4100 and NZS 3404.
- Plotting LTB restraints in OpenGL view.
- Member steel design report accessible from right-click.
- Box sections checked by Limsteel (AS 4100 and NZS 3404).
- Monosymmetric I sections checked by Limsteel.
- Allows tee section design strength increase for stem in tension.

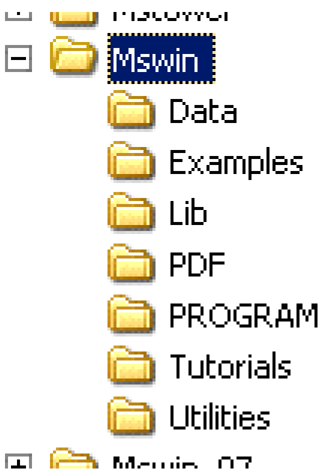
2:Getting Started

Installing Microstran

The Setup program will install Microstran on your computer. You must have Administrator privileges to install Microstran.

Place the Microstran CD into your drive and the installation will start automatically. Setup will guide you through the installation process, prompting you as required.

A number of folders will be established under the specified Microstran folder. If you use the default name the folders as displayed in Windows Explorer will look like this:



Several tutorial examples are included in Chapter 12 – “Input Tutorials” so that you can easily go through all the steps needed to do a typical job. Some of the tutorial examples are available as self-playing demo files that can be viewed on your computer. These files are on the Microstran CD and can also be downloaded from the Microstran website.

Folder Name	Comment
Mswin	Microstran folder – you can choose this name during installation. Mswin is the default.
.....Data	Default data folder – you can open Microstran files in other folders if you wish.
.....Examples	Example and tutorial files – useful for testing and learning.
.....Lib	Library files, design parameter files, and template files.
.....PDF	Microstran User Manual in PDF format.
.....Program	All Microstran program files and help files.
.....Tutorials	Created optionally during installation, this folder contains “self-playing demos”.
.....Utilities	Programs to help convert data and section libraries from old versions of Microstran.

Hardware Lock

Microstran is normally supplied with a hardware lock that must be attached to a USB port before you can start the program. No drivers are required. Additional set-up procedures are required for systems with a network lock. These are described on a data sheet supplied with the software.

Starting Microstran

The Setup program creates a Microstran icon on the Windows All Programs menu. Select **Start > All Programs > Microstran** to start Microstran.

Note the following help features, which make it easier for you to use Microstran:

- There are **tooltips** on all toolbar buttons. Move the mouse cursor over the button for a moment and a little pop-up window displays the function of the button.
- There is a prompt displayed on the left side of the status bar (at the bottom of the Microstran window) whenever the cursor is positioned over a toolbar button or a menu item. *Look here for prompts while you are performing input operations.*
- Context-sensitive (pop-up) help is available in most dialog boxes. Some items in dialog boxes also have tooltips.

- The Command Assistant, available for most commands, displays a pop-up window listing the necessary steps.

Use the **Help > Microstran Help Topics** command to display the Help Topics dialog box. With this, you can browse the table of contents, look through an index, or search all help topic keywords.

Font

Microstran requires the font MS LineDraw. During installation this font is installed automatically using the file Linedraw.ttf on the distribution CD. When setting up Microstran on a workstation without installing from the CD you must ensure that MS LineDraw is installed.

If unusual characters are seen in Print Preview it could mean that MS LineDraw is not properly installed or the system has not been rebooted after installation of the font.

How to Make a Shortcut on the Desktop

To make a shortcut to Microstran on your *desktop* (the background that is visible when no programs are running), right-click on the desktop, select **New > Shortcut**, and in the Create Shortcut dialog box browse to the Msw9.exe file in the Microstran \Program folder. Set the "Start in" folder to the data folder. Enter **Microstran** for the name of the shortcut, and click the **Finish** button. Alternatively, drag the Microstran icon from the Start menu to the desktop with the mouse while pressing the Ctrl key.

Launch with Double-Click

Microstran job files (Job.msw, where "Job" is the job name) should be identified in Explorer with the icon,  . It is convenient to be able to double-click on one of these files in Explorer to start Microstran with the job. To do this, the .msw file type must be associated with Microstran.

The association between Microstran and the .msw file type may be established when Microstran is installed. If lost, the association may be re-established with the procedure set out below.

Here are the steps necessary to make Microstran launch with a double-click:

- In Explorer select the **View > Folder Options** or **View > Options** command.
- Select the File Types tab.

- In the list box search for the Microstran job file type, which may be shown as either “Mswin Document” or “MSW File”. If found, select this file type and click the Remove button. Close the dialog box.
- In Explorer browse to the data folder and double-click on any Microstran job file (if the file name extension “msw” is not visible you may see it by right-clicking and checking the properties of the file).
- The Open With dialog box appears. Click on the Other button and browse to Msw9.exe in the Microstran \Program folder.
- In the Description box type “Microstran Job File” and click OK.
- In Explorer select the **View > Folder Options** or **View > Options** command.
- Select the File Types tab, then select “Microstran Job File” in the list box and click the Edit button.
- Click the Change Icon button and then select the second icon.
- Click OK to close the Edit File Type dialog box.
- Click OK to close the Folder Options dialog box.

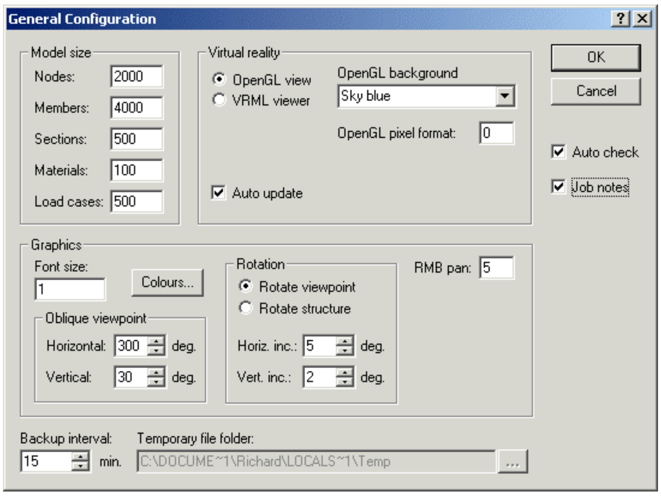
Now, check that you have successfully set up your system by browsing to a Microstran job file and double-clicking.

Configuration

Microstran may be configured in many different ways according to your preferences. Configuration settings are saved from run to run, for each user. The **File > Configure** command allows you to set items such as font size, colours, and default library files and design codes. Window location and size and toolbar locations are saved as they are when Microstran closes.

Microstran configuration settings are saved in the Windows registry under the HKEY_CURRENT_USER key. When you start a new job configuration settings remain as they were at the last run. You may reset configuration settings to their default values by running the MsReset.exe program in the Microstran \Program folder.

The dialog box shown below is for general configuration items. Pop-up help is available for each item by clicking the button and then clicking the item.



FILE > CONFIGURE > GENERAL

Auto check

With this box checked Microstran automatically attempts to check from the internet whether the main executable file is the latest available. This check occurs only on the first run for any day. With this box checked Microstran may also attempt to check from the internet whether maintenance is current.

Job notes

With this box checked Microstran displays a window containing notes specific to the job that have been input by the user. The notes window may be closed at any time and restored with the **Structure > Notes** command.

Model size

Microstran will allocate sufficient memory for a model of the size specified. Setting values higher than necessary may lead to a loss of efficiency and reduce the amount of memory available for storing results. *Any change will take effect the next time you start Microstran.*

OpenGL pixel format

On computers supporting OpenGL there are 24 or more *pixel formats*, only some of which are suitable for Microstran's virtual reality view. When the pixel format in the OpenGL Configuration dialog box is set to zero a suitable pixel format is chosen automatically. If your display has the necessary color depth OpenGL does not work, you may need to set the pixel format. Contact Microstran support if you need help choosing an appropriate pixel format.

WARNING: Use of an inappropriate value may crash your system.

Auto update

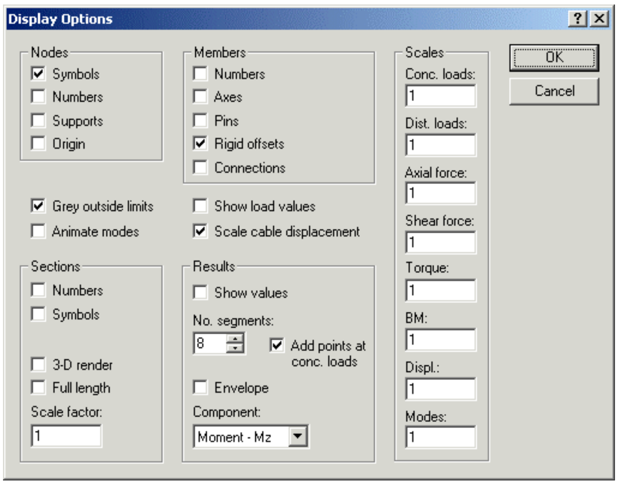
With this box checked Microstran automatically updates the OpenGL view every time the structure is changed. When working with a very large model responsiveness may be improved by clearing this box.

RMB pan

With this value greater than zero you may pan the main view by clicking and dragging with the right mouse button. The pop-up menu will then appear only with the mouse stationary when you right-click. Ensure this option is not selected if you want to initiate the pop-up menu with the mouse moving. The value represents the number of pixels of movement detected before panning is triggered.

Display Options

The **View > Display Options** command may be used to change display settings but most items in the dialog box can be set more easily by using buttons on the Display toolbar.



VIEW > DISPLAY OPTIONS

No. segments

Member force diagrams are plotted at a number of internal points in each member determined by this value. The default value is 8. For large structures it may be necessary to reduce this value to 4 or even 1 in order to allow sufficient memory for the storage of results.

Add points at conc. loads

With this setting extra points are allocated for computation of results at locations where point loads occur.

Microstran Commands

Microstran commands may be performed by selecting an item on a drop-down menu, clicking a toolbar button, or right-clicking and then selecting an item on the context menu. Almost all the commands are available on the drop-down menus while some of them are also available on toolbar buttons or the context menu. Commands selected from the drop-down menus are referred to in this manual as shown in this example:

View > Zoom > Window

Commands selected by clicking a toolbar button are referred to by the name of the button, as shown in the tooltip.

Most commands operate on one or more nodes or members. You may use either of these command formats:

Command-Then-Select

- Select a command on a drop-down menu or click a toolbar button.
- Pick the nodes or members.
- Right-click and choose OK on the context menu.

Select-Then-Command

- Pick the nodes or members.
- Select a command on a drop-down menu, click a toolbar button, or right-click and choose a command on the context menu.

Shortcut Keys

Microstran permits the use of *shortcut keys* to some commands. Shortcut keys are also known as *accelerator keys*. The effect of pressing a shortcut key depends on the context. For example, pressing Delete usually deletes selected members, but in a dialog box it may delete text or do nothing. The shortcut keys used to control Microstran's main view are listed below:

Shortcut	Command
←	Viewpoint left *
→	Viewpoint right *
↑	Viewpoint up *
↓	Viewpoint down *
Page Up	Zoom in
Page Down	Zoom out
Ctrl+Z	Undo
Ctrl+Y	Redo

F5	Redraw
Ctrl+A	Select all
Shift	Enter Dynamic Rotate mode
Delete	Erase members
Home	Zoom to extents/limits
Esc	Cancel
Enter	Confirm or OK
Space	Display next case/mode

* May be configured to rotate structure instead of moving viewpoint.

These standard Windows shortcuts may be used in dialog boxes.

Shortcut	Command
Ctrl+C	Copy
Ctrl+X	Cut
Ctrl+V	Paste
Ctrl+Z	Undo
Ctrl+Y	Redo
Ctrl+A	Select all (in list box)
Delete	Delete character
Home	Move caret to beginning of field
Esc	Cancel
Enter	OK

See "Shortcut Keys" on page 122 for shortcuts used in Graphics Input and the OpenGL virtual reality view.

Using the Mouse

Mouse functions are used in Microstran's main view to perform these operations:

Function	Operation
Click	Select node or member
Right-click with mouse stationary	Display pop-up menu
Double-click	Node or member properties
Click & drag left button	Select nodes or members with "rubber band" box
Click & drag middle button	Pan
Click & drag right button with mouse moving*	Pan
Roll wheel	Zoom view

* The sensitivity may be adjusted in General Configuration.

These mouse functions are used in the OpenGL view:

Function	Operation
Right-click with mouse stationary	Display pop-up menu
Double-click	Re-centre on clicked point
Click & drag middle button	Pan
Click & drag right button with mouse moving	Pan
Roll wheel	Zoom view

Right Mouse Button Pan

You may pan the view by clicking and dragging with the right mouse button. If the mouse is stationary the pop-up menu is displayed; otherwise, the Pan command is initiated. In General Configuration you may adjust the amount of mouse movement required before the Pan command is triggered in the main view. See **RMB pan**, above.

Selecting Nodes and Members

Key concept.

You may select nodes or members in several ways:

- Clicking each node or member in turn. Clicking again on a node or member deselects it.
- Dragging a selection box that encloses the nodes or members to be selected. “Dragging a selection box” means clicking (with the left mouse button) a point away from the nodes or members to be selected, then dragging the mouse until the selection box encloses the necessary nodes or members, and finally, releasing the left mouse button. Note that when the selection box is dragged from right to left, a “crossing window” appears, which selects not only members enclosed by the box but also members cut by the sides of the box.
- All members may be selected by Ctrl+A (see “Shortcut Keys”, above).
- Selecting a predefined set.

Selected members are displayed with a dotted line while selected nodes are shown enclosed in a small square.

Microstran lets you store a selection of nodes or members in a *set* (see “Sets”, below). This is particularly useful with large or complex models, simplifying the task of reselecting a group of nodes or members every time you need to operate on them.

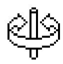
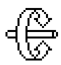
Cursors

Key concept.

Microstran displays various cursors at different times, depending upon what is happening. These cursors are shown in the table below.

Generally, when you have finished a command, Microstran allows you to repeat the command until you *cancel* the command by right-clicking. For example, when you select the **Structure > Erase Members** command, the cursor changes, you then select members you want to erase and confirm the selection by right-clicking and choosing **OK** on the context menu. The member selection cursor is still displayed, allowing you to choose more members to erase. To terminate the command, right-click, and the standard arrow cursor will reappear.

Many commands are *interruptible*. This permits you to adjust the view during a command. When drawing members in a large model, for example, having clicked the “A” node of a member, you may need to zoom in to another region of the structure before clicking the “B” node.

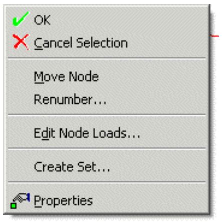
Cursor	Description
	Command mode. Microstran is waiting for you to select a command from the menu, click a toolbar button, or select a node or member (the cursor changes as soon as you select a node or member).
	Drawing mode. Microstran is waiting for you to click an end of a member. Look at the right of the status line to determine which snap mode is in effect. You may use the Structure > Drawing Settings command or the context menu to change the snap mode without leaving the current drawing command.
	Member selection mode. Microstran is waiting for you to select one or more members by clicking on them or enclosing them in a selection box. If you drag a selection box from left to right, cut members are excluded. Dragging from right to left includes cut members.
	Node selection mode. Microstran is waiting for you to select one or more nodes by clicking on them or enclosing them in a selection box.
	This cursor appears when you are selecting a zoom window or you are about to commence dynamic rotation. When zooming, drag from one corner to the diagonally opposite corner of the rectangle you want to zoom to. When using the Dynamic Rotate command, click and drag to rotate the viewpoint (or structure).
	This cursor appears when you click and drag the mouse horizontally while you are using the Dynamic Rotate command. It indicates that the viewpoint (or structure) is being rotated about a vertical axis. This mode is cancelled when the mouse button is released.
	This cursor appears when you click and drag the mouse vertically while you are using the Dynamic Rotate command. It indicates that the viewpoint (or structure) is being rotated about a horizontal axis. This mode is cancelled when the mouse button is released.
	Dynamic zoom mode. Microstran is waiting for you to click and drag the mouse up (zoom in) or down (zoom out). This mode is cancelled when the mouse button is released.
	Dynamic pan mode. Microstran is waiting for you to click and drag the mouse to move the image. This mode is cancelled when the mouse button is released.

Right-Clicking on Nodes and Members

Key concept.

Microstran fully implements the Windows protocol for right-clicking on objects to obtain a menu of related commands – the context (or pop-up) menu. The Pan command will be initiated if the mouse is moving when you right-click. You may adjust the amount of movement required to trigger the Pan command in General Configuration.

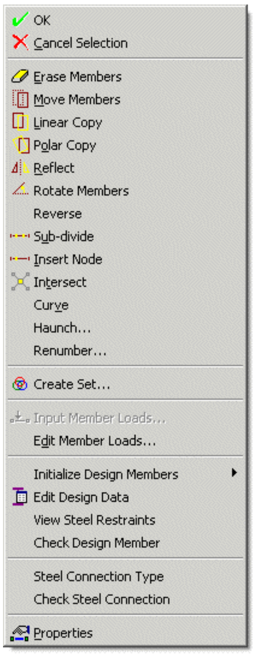
Right-clicking on a node displays this context menu:



NODE CONTEXT MENU

*Double-clicking on a node is the same as selecting **Properties** on this pop-up menu.*

This is the context menu that appears when you right-click on a member:

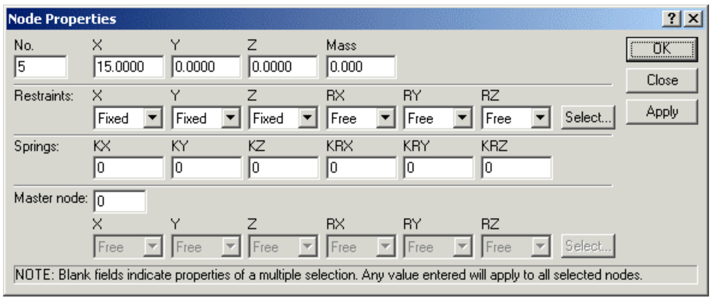


MEMBER CONTEXT MENU

*Double-clicking on a member is the same as selecting **Properties** on this menu.*

The Node Properties Dialog Box

The dialog box shown below appears when you double-click a node or select **Properties** after right-clicking a node. This provides the simplest way to edit any of the node properties.

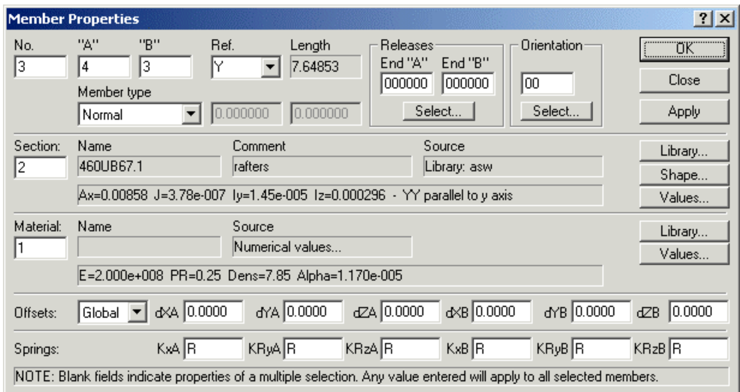


NODE PROPERTIES DIALOG BOX

Note: You may click on any other node and the contents of the dialog box will be updated for the new node.

The Member Properties Dialog Box

The dialog box shown below appears when you double-click a member or select **Properties** after right-clicking a member. This provides the simplest way to edit any of the member properties.



MEMBER PROPERTIES DIALOG BOX

Note: You may click on any other member and the contents of the dialog box will be updated for the new member.

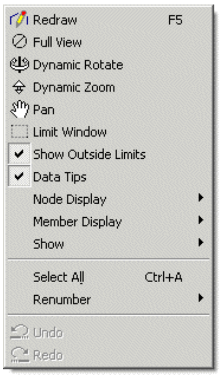
Properties Dialog Boxes with Multiple Selection

Key concept.

You may select several nodes or members, then right-click and choose **Properties** on the context menu. The dialog box will display *common properties* of the selected group of nodes or members. Blank edit boxes indicate that the corresponding value is not the same for all of the multiple selection. Any changes you make will be applied to the whole group when you click **Apply** or **OK**.

Right-Clicking Away from Any Node or Member

When you right-click in the main window, away from any node or member, the pop-up menu below appears.



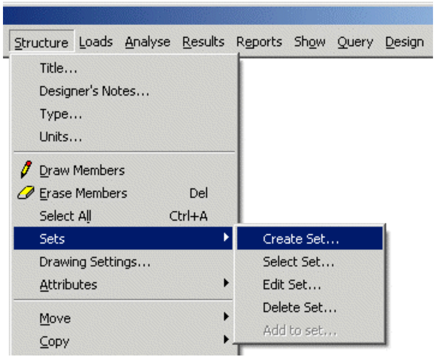
MAIN CONTEXT MENU

This provides a very convenient alternative to the drop-down menus for many commands. In effect, you can perform some operations in three different ways. For example, you can display the section number on all members by clicking a button on the Display toolbar, by selecting the **View > Display Options** command, or by right-clicking and then selecting **Section Numbers**.

Sets

A set is a group of structure entities that may be referred to by name to facilitate selection. Sets are stored with the job so that once you have defined them they can be selected in subsequent runs. *Renumbering the structure does not change the set.*

Sets are managed with the **Structure > Sets** sub-menu, shown below.

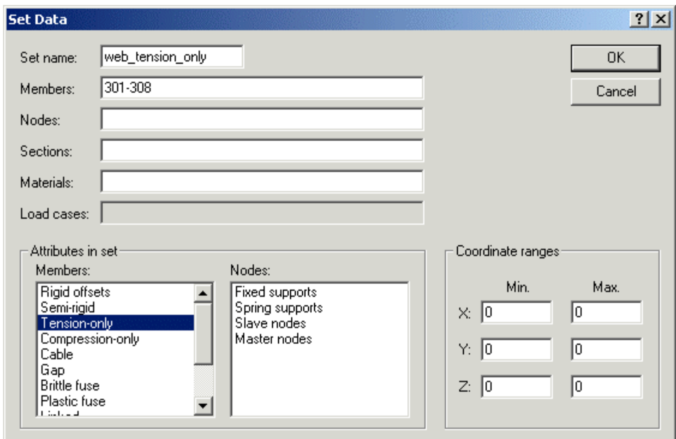


STRUCTURE > SETS SUB-MENU

Create Set

This command displays the dialog box below, in which you may assign a set name and specify lists of entities in the set. The set name goes into the combo box on the View toolbar along with the names of any other defined sets. Selecting the nodes or members in the set is then as simple as selecting the name of the set in the combo box drop-down list.

The Create Set command is also available on the node and member context menus. After selecting nodes or members you may right-click and choose Create Set from the pop-up menu.



SET DATA DIALOG BOX

Members are included in the set if they have all the selected member attributes and their centre lies within the coordinate ranges. Nodes are included in the set if they have all the selected node attributes and they lie within the coordinate ranges.

Create Member Set

On selecting this command (also accessible from a button on the Extra toolbar) you may select members for inclusion in a new set. On confirming the selection on the context menu the Set Data dialog is displayed so you can enter the set name. You may now cancel the command with a right-click or proceed to select the members in another new set.

Select Set

This command selects nodes and members in the selected set. The same command is initiated when you select a set from the combo box list on the View toolbar.

Edit Set

On selecting this command you may select a set from a list of all sets in the Select Set dialog box and then you may change any item for this set in the Set Data dialog box.

Delete Set

This command lets you select a set to be deleted.

Printing in Microstran

Print and Print Preview Commands

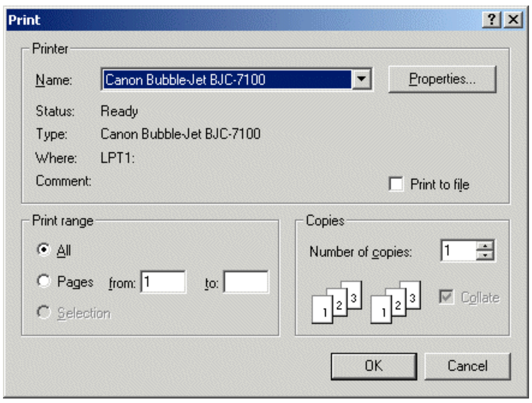
Microstran differs from many Windows application in that there is a requirement to print both files (reports) and pictures. As in a standard Windows application, Microstran has a Print command on the File menu (**File > Print File**). This is for printing files and reports. Also, there is a Print command on the View menu (**View > Print View**) and this is used for printing pictures of the structure.

In addition to Print commands on the File and View menus, Microstran has Print Preview commands on each of these menus. The print preview shows an exact image on the screen of the printed page. **File > Print Preview** shows you how a report will be printed while **View > Print Preview** is for Microstran graphics.

The main toolbar, usually located right under the menu, contains a Print button,  , and a Preview button,  . These buttons are for Microstran graphics, not files or reports. They correspond to the Print and Preview commands on the View menu. The main toolbar is shown in "Main Toolbar Commands" on page 52.

The Windows Print Dialog Box

The **File > Print File** and **View > Print View** commands display the Windows Print dialog box so you can change the target printer, the number of copies, or printer settings immediately before printing. When you click OK in this dialog box the selected printer becomes the current printer. Clicking the Print button on the main toolbar initiates a graphics print *without* the display of the Windows Print dialog box. The view is printed immediately to the current printer – notice that the tooltip for the Print button shows the name of the current printer.



WINDOWS PRINT DIALOG BOX

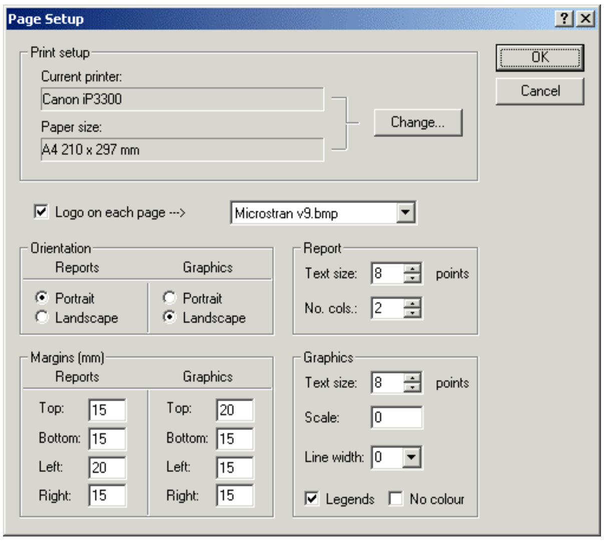
Note: Clicking the Properties button displays the printer properties dialog box. The page orientation setting in the printer properties dialog box is ignored. Microstran maintains separate page orientation setting for graphics and reports.

Preview commands, **File > Print Preview**, **View > Print Preview**, and the Preview button, all do not display the Windows Print dialog box. The preview is always for the current printer. When you see a print preview on the screen there is a Print button at the top left of the preview window. Clicking this will initiate printing on the current printer. *If you want to change the target printer after seeing a preview, close the preview window and then select the Print command on either the File or the View menu.* When previewing a multi-page report file, the Print button prints the whole file. *If you want to print less than the full report use the* **File > Print File** *command and select the pages to be printed in the Windows Print dialog box.*

The Page Setup Dialog Box

The Page Setup dialog box allows you to change settings affecting the layout of printed output, either graphical or reports.

The current printer, shown in the Page Setup dialog box, is initially the Windows default printer and remains so until a different printer is selected. A new current printer may be selected in the Windows Print Setup dialog box that is shown when you click the Change button. You may also change the current printer in the Windows Print dialog box shown when you select either **View > Print View** or **File > Print File**.



MICROSTRAN PAGE SETUP DIALOG BOX

Logo

Check this box if you want Microstran to print a logo at the top of each page of printed output. When the box is checked you may choose one of the available bitmap files from the adjacent combo box. See "Configurable User Graphic" on page 36.

Orientation

Microstran does not use the orientation setting stored with the printer properties. These two settings, one for reports and one for graphics, are used instead.

Margins

Margins may be set independently for reports and graphics.

Report

Text size	The point size of the text used for printing – typically 7 or 8 points. To make very compact reports you may select a smaller value, particularly if you also select two or more columns.
No. cols.	You may select more than one column for the printing of reports. The number of columns that will fit on a page depends on the size of the paper, the orientation, and the size of the text. If you select more columns than will fit on the page, the number of columns is automatically reduced. If the text size is so large that not even one column will fit, a warning is displayed and the lines are truncated.

Graphics

Text size	The point size of the text used for printed graphics – typically 7 or 8 points.
Scale	The scale at which structure geometry is shown. With a scale of 100, for example, 1 m on the structure is represented as 10 mm on the plot. When the scale is zero (default) the structure is plotted to fill the space available.
Line width	Choose a non-zero value from the drop-down list box for thicker lines in graphical output.
Legends	Colour legends for sections and load cases may be shown. The section legend is only shown when section numbers are included on the plot. The load case legend is only shown for the load cases for which loads are plotted.
No colour	With the exception of the configurable user graphic, which is always printed in its own colours, printing is in black only, even if using a colour printer.

Configurable User Graphic

You may use this feature to place your company logo at the top of all printed output.

Microstran allows you to have a small graphic at the top of each page of printed output. Any valid Windows bitmap file existing in the Microstran \Program folder may be selected in the Page Setup dialog box. With this option selected the graphic is printed on each page. If the option is not selected no graphic will be printed and no space will be allowed for it. On installation Microstran is configured to use the graphic shown below. You can unselect the option in Page Setup if you do not want a graphic.

Microstran V9

DEFAULT GRAPHIC

The specification of the bitmap is:

- Width – 1200 pixels
- Height – 200 pixels
- Colours – 256

Bitmaps that do not match these requirements are not shown in the Page Setup dialog box. Microstran prints the graphic in a space 50.8 mm wide by 8.5 mm high.

Note: The Windows drivers for some printers do not support the printing of bitmaps.

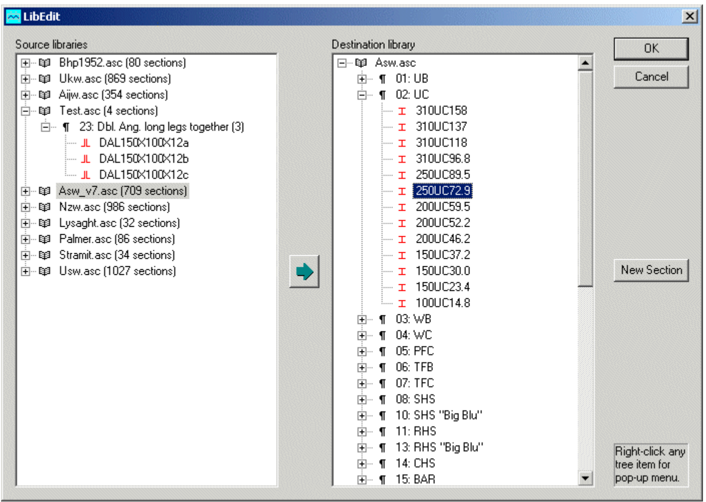
Steel Section Libraries

Microstran section libraries have a file name extension of “lib” and reside in the library folder. A source file is usually supplied with each steel section library. The source file is a text file with the file name extension “asc”, which containing manufacturers’ section data.

The **Configure > Section Library Manager** command gives access to powerful facilities for editing an existing library or making a new library by merging sections and section categories from existing libraries – see Chapter 18 – “Section & Material Libraries”. When a library is saved it may be compiled into a library file accessible to Microstran (see “Compiling a Library” on page 313).

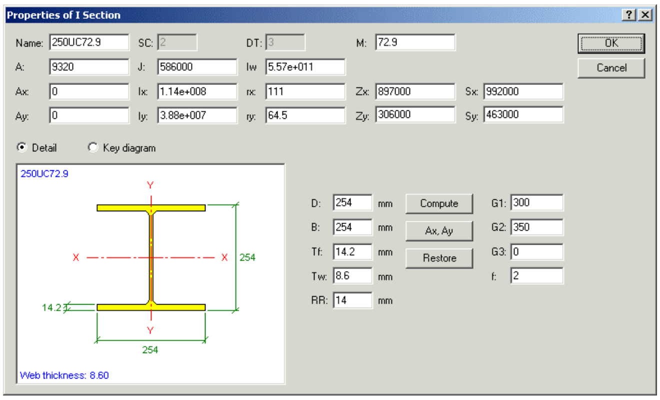
It is recommended that you do not modify the standard library supplied with Microstran – it is preferable to copy it to a file with a different name and then modify that.

The Section Library Manager window shows a tree view of all available libraries on the left and a tree view of the destination library on the right. Any library, section category, or section may be sent from the left tree to the right tree by selecting it and then clicking the arrow button in the centre.



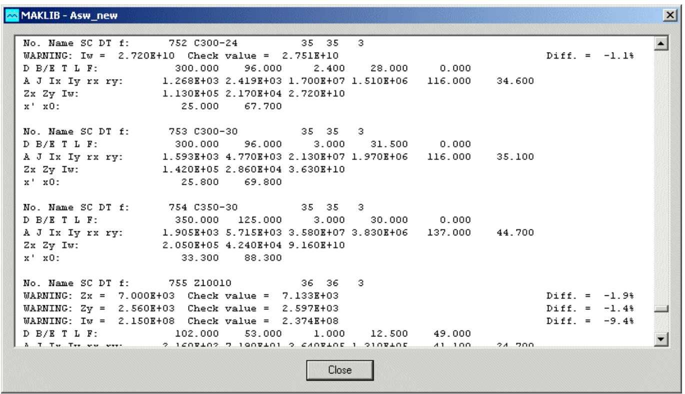
SECTION LIBRARY MANAGER

Double-clicking a section in the right-hand tree displays a dialog box that lets you change any value stored in the library. The Compute button displays derived section properties calculated by Microstran.



SECTION PROPERTIES

Compiling the saved library source file produces a report that shows any difference of 1% or more between original derived values and those computed by Microstran.



COMPILING THE LIBRARY

Data from Earlier Versions

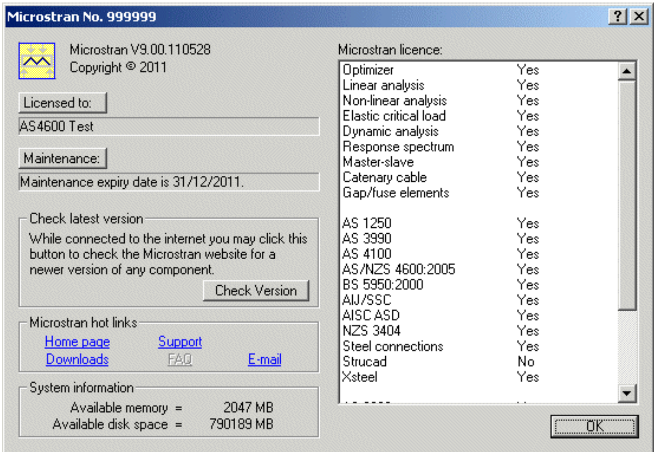
Microstran V9 reads .msw files from V8, V7, and V6.5. It can also import data in archive file format written by earlier versions of Microstran. To import an archive file use the **File > Import > Archive File** command.

If any error messages are displayed during the importing of the archive file it will be necessary to edit the file (**File > List/Edit File** command).

Note: Previous versions of Microstran cannot read files saved by Microstran V9 but may be able to import a V9 archive file. Editing the archive file may be necessary.

Technical Support

Microstran technical support is available to eligible users by e-mail. Use the **Help > About Microstran** command to display the serial number, the version number, and licence details for your software. This information is required when you ask for technical support. The Help About dialog box contains links to the Microstran website, where you may search FAQ topics for help, submit a support request, or check that your software is up to date – see “Web Update” on page 40.



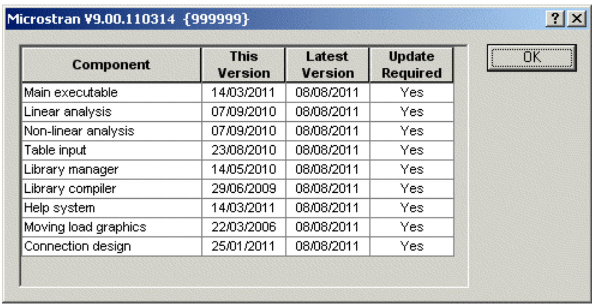
HELP ABOUT MICROSTRAN

Maintenance

Maintenance is an optional arrangement under which software updates and support are provided. There is an initial free maintenance period of 12 months from the date of purchase or 6 months from the date of purchase of an upgrade from Microstran V8. The maintenance expiry date is shown in the Help About dialog box. Full support services are available to users with a maintenance subscription. Basic support is available to users without a maintenance subscription, depending on availability of resources. Contact your dealer or Engineering Systems to purchase maintenance renewal.

Web Update

From time to time, updates may be available to eligible users on the Microstran website. While your computer is connected to the internet, clicking the Check Version button in the Help About dialog box displays the dialog box shown below. This shows the dates of your Microstran software and dates of the current web downloads, making it easy to see whether an update is required.



Component	This Version	Latest Version	Update Required
Main executable	14/03/2011	08/08/2011	Yes
Linear analysis	07/09/2010	08/08/2011	Yes
Non-linear analysis	07/09/2010	08/08/2011	Yes
Table input	23/08/2010	08/08/2011	Yes
Library manager	14/05/2010	08/08/2011	Yes
Library compiler	29/06/2009	08/08/2011	Yes
Help system	14/03/2011	08/08/2011	Yes
Moving load graphics	22/03/2006	08/08/2011	Yes
Connection design	25/01/2011	08/08/2011	Yes

MICROSTRAN WEB UPDATE DIALOG BOX

You can connect to the Microstran website by clicking the Downloads hot link in the Help About dialog box. Here, you will recognize the components you need to download. Each download is an executable file – run it to unpack the update files. If prompted for a password when this executable runs you must e-mail Microstran Support to obtain it.

Tutorials & Examples

If you are new to Microstran it is recommended that you work through one or more of the tutorial examples in Chapter 12 – “Input Tutorials” to familiarize yourself with the operation of the principal menu and toolbar items. On installation you have the option of installing a number of “self-playing demos”. Each of these consists of an executable file in the tutorials folder. In Windows Explorer, you may double-click on any one of them to see the tutorial.

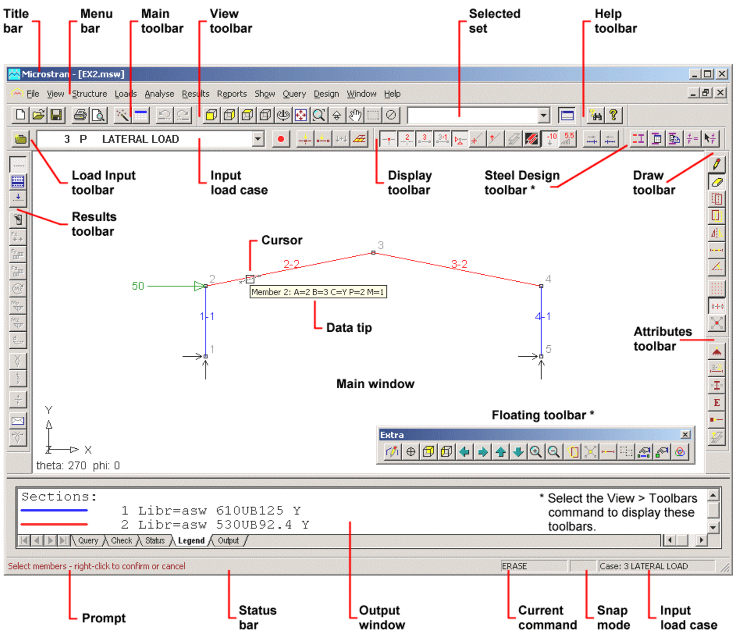
The Examples folder contains a number of example jobs, which are referred to in Chapter 20 – “Examples”. Use the **File > Open** command to run one of these jobs.

3:Menus & Toolbars

Layout

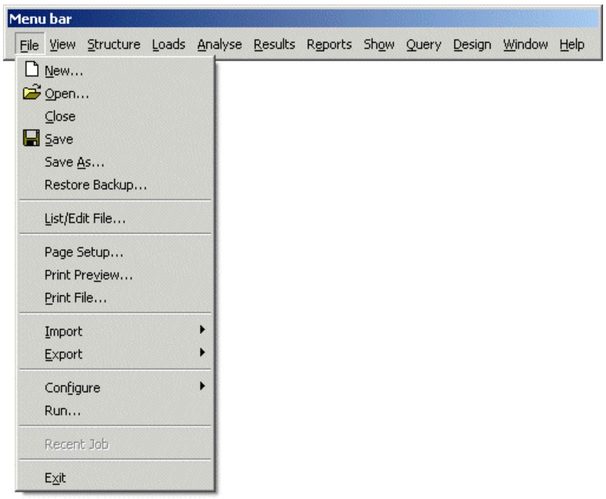
The diagram below shows the layout of the Microstran window. Commands may be initiated from the *menu bar*, any toolbar, or a context (pop-up) menu. Each *menu title* gives access to a *drop-down menu*. Some items on drop-down menus lead to *sub-menus*. Each toolbar button usually corresponds to a command on a drop-down menu. All commands that appear on a toolbar have an associated icon, which is displayed when the same command appears on a menu. This chapter lists all the commands available from the menu bar and all toolbars.

Note: See Chapter 2 – "Getting Started" for detailed information on the pop-up menus that appear when you right-click with the mouse.



LAYOUT OF MICROSTRAN WINDOW

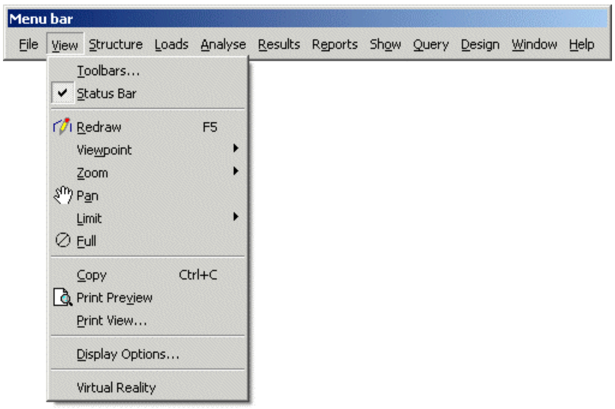
File Menu Commands



FILE MENU

Command	Action
New	Creates a new job.
Open	Opens an existing job.
Close	Closes the current job.
Save	Saves the current job using the same file name.
Save As	Saves the current job to a specified file name and changes the name of the current job accordingly.
Restore Backup	Reinstates the current job as it was at the time of the last automatic backup.
List/Edit File	Opens the selected file with the Microstran text editor for viewing or editing.
Page Setup	Change the printing options.
Print Preview	Displays the selected file on the screen, as it would appear printed.
Print File	Prints the selected file.
Import	Reads data into Microstran from a file (e.g. Microstran Archive file).
Export	Writes Microstran data to a file for input to another program. Also used for saving job in the form of a Microstran Archive file.
Merge Models	Merge archive files for two models into a new archive file.
Configure	Configuration of program capacity, section library, material library, colours, intermediate file folder, and timed backup interval. Also used for editing of section and material libraries and dynamic response spectra.
Run	Run another program.
Recent Job	Selects recently used job.
Exit	Exits Microstran

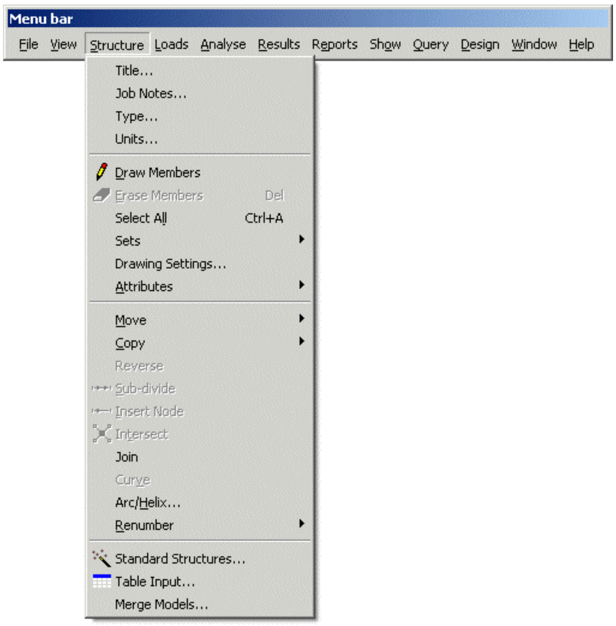
View Menu Commands



VIEW MENU

Command	Action
Toolbars	Shows or hides the toolbars.
Status Bar	Shows or hides the status bar.
Redraw	Redraws the current view.
Viewpoint	Change the orientation of the structure in the view.
Zoom	Change the scale of the view or select a rectangular part of the view to fill the display window.
Pan	Displace the view by the selected distance.
Limit	Restrict input to a part of the structure by one of several available methods. Parts outside the limits are shown in light grey or hidden.
Full	Makes whole of structure available for input. This reverses the effect of the Limit command.
Copy	Copy view to Windows clipboard in EMF format.
Print Preview	Displays the view as it would appear printed.
Print View	Prints the view.
Display Options	Select options for displaying node numbers, member numbers, etc.
Virtual Reality	Create a virtual reality view of the structure using OpenGL or VRML, according to configuration.

Structure Menu Commands

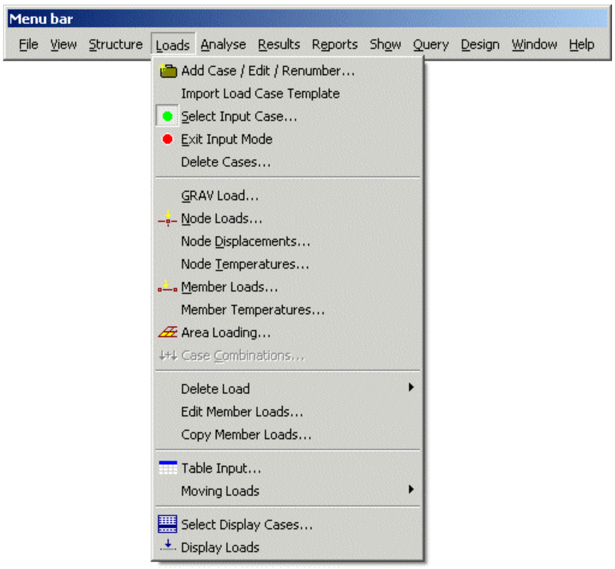


STRUCTURE MENU

Command	Action
Title	Input job description.
Job Notes	Input notes relating to the job.
Type	Specify structure type (plane frame, space frame, etc.).
Units	Specify system of units.
Draw Members	Draw members or input node coordinates.
Erase Members	Erase selected members.
Select All	Selects all members, including any that may not be visible.
Sets	Create, select, edit, or delete a set. A set is a named group of entities forming any part of the structure.
Drawing Settings	Snap modes for drawing members, grid spacing etc.
Attributes	Input attributes of the structure, such as restraints, section numbers, etc.
Move	Move a node, move members, rotate members, stretch nodes.
Copy	Linear copy, polar copy, reflect members.
Reverse	Reverse the A and B nodes of the selected members.
Sub-divide	Sub-divide selected members into a number of equal parts.

Insert Node	Insert a new node in a member.
Intersect	Insert new node(s) at intersection of selected members.
Join	Replace contiguous collinear members by a single member, preserving loads and design data.
Curve	Sub-divide a member into a number of segments whose ends lie on an arc.
Arc/Helix	Create members with ends lying on arc or helix.
Renumber	Renumber nodes and members (sort or compact).
Standard Structures	Input structure by selecting from common types.
Table Input	Spreadsheet style of input.
Merge Models	Create new model from two existing models.

Loads Menu Commands

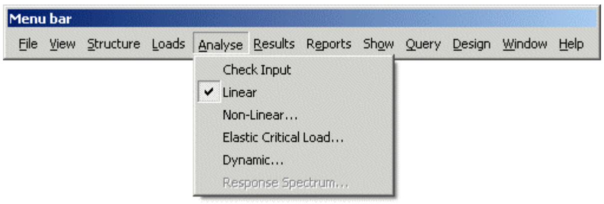


LOADS MENU

Command	Action
Add Case / Edit Title	Add new load case or edit title of existing load case.
Import Load Case Template	Add load cases specified in a load case template file.
Select Input Case	Select load case for input of loads.
Exit Input Mode	Deselects input load case.
Delete Case	Delete load case.
GRAV Load	Input acceleration.

Node Loads	Input loads at selected nodes.
Node Displacements	Input prescribed displacement at specified nodes.
Node Temperatures	Input temperature at selected nodes.
Member Loads	Input loads on specified members.
Member Temperatures	Input temperature (gradients) in selected members.
Area Loading	Input load per unit area on selected area.
Case Combinations	Input combination load case.
Delete Load	Delete selected type of load in selected load case.
Edit Member Loads	Change loads on selected members.
Copy Member Loads	Copy member loads from source member to target members.
Table Input	Spreadsheet style of input.
Moving Loads	Generate load cases for moving loads.
Select Display Cases	Select cases for display of loads or results.
Display Loads	Display applied loads for selected cases.

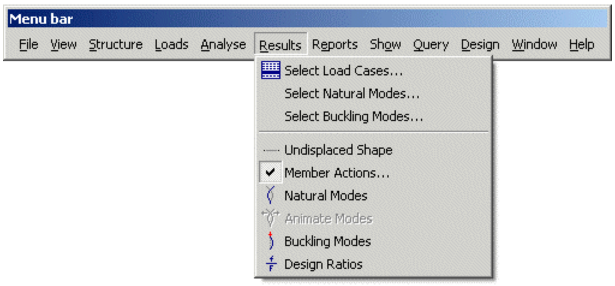
Analyse Menu Commands



ANALYSE MENU

Command	Action
Check Input	Check structure and load data (normally automatic).
Linear	Perform linear analysis (first-order).
Non-Linear	Perform non-linear analysis (second-order).
Elastic Critical Load	Determine frame buckling load factors and buckling mode shapes.
Dynamic	Determine natural frequencies and mode shapes.
Response Spectrum	Add response spectrum and static analysis results.

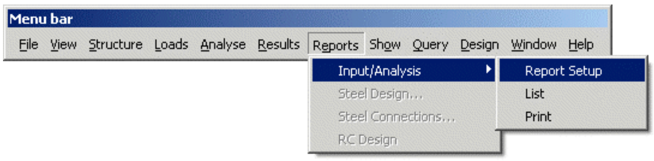
Results Menu Commands



RESULTS MENU

Command	Action
Select Load Cases	Select load cases for display of loads or results.
Select Natural Modes	Select modes for display of vibration mode shapes.
Select Buckling Modes	Select modes for display of buckling mode shapes.
Undisplaced Shape	Display structure in undisplaced position.
Member Actions	Display bending moment, shear force, axial force, torque, or displaced shape.
Natural Modes	Display vibration mode shapes.
Animate Modes	Show each currently displayed mode (natural or buckling) in alternate extreme positions. Press the space bar to show the next mode, Esc to cancel.
Buckling Modes	Display buckling mode shapes.
Design Ratios	Display results of member design check with colours representing range of design ratios. The legend in the output window shows the range of values represented by each colour.

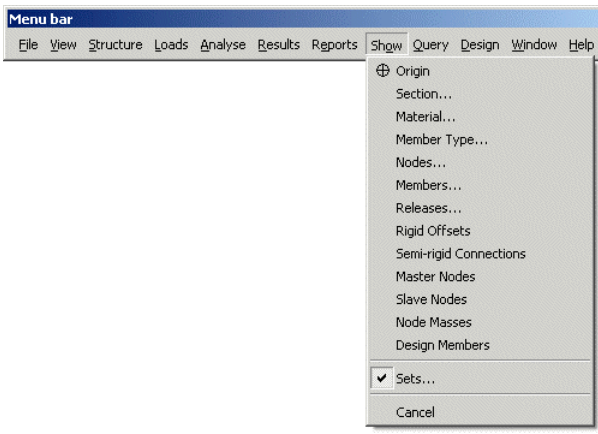
Reports Menu Commands



REPORTS MENU

Command	Action
Input/Analysis	Input/analysis report – select option for setup, list, or print.
Steel Design	Create design report.
Steel Connections	Create steel connection report.
RC Design	Not required – report is created automatically.

Show Menu Commands

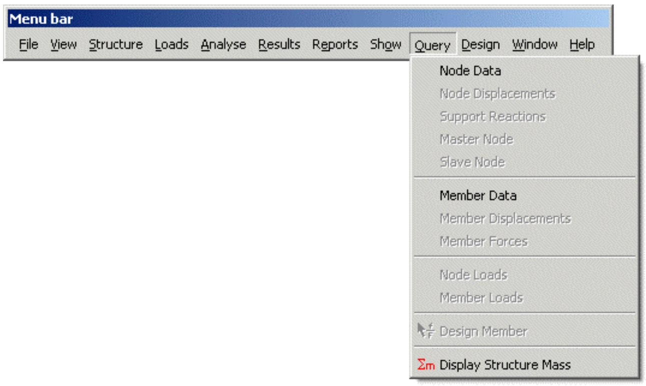


SHOW MENU

Command	Action
Origin	The origin of the global coordinate system is shown with a cross inside a circle.
Section	Highlight members with specified section number.
Material	Highlight members with specified material number.
Member Type	Highlight members of specified type (tension-only etc.).
Nodes	Highlight members connected to specified nodes.

Members	Highlight specified members.
Releases	Highlight members with specified type of release.
Rigid Offsets	Highlight members with rigid offsets.
Semi-rigid Connections	Highlight members with semi-rigid connections.
Master Nodes	Highlight master nodes.
Slave Nodes	Highlight slave nodes.
Node Masses	Highlight all nodes with non-zero added mass.
Design Members	Highlight initialized design members.
Steel Connections	Highlight steel connections.
Sets	Highlight specified set.
Cancel	Cancel current “Show” selection.

Query Menu Commands



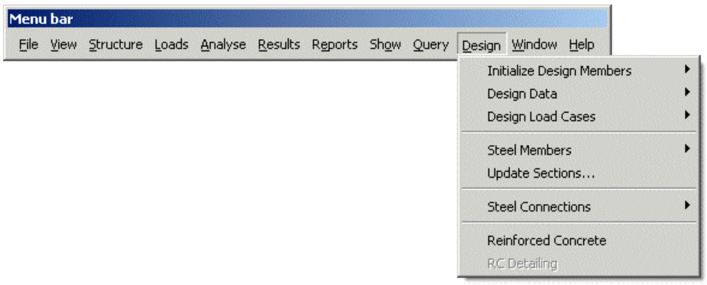
QUERY MENU

Command	Action
Node Data	List data for selected node (coordinates, restraint, etc.).
Node Displacements	List displacements for selected node.
Support Reactions	List reactions for selected (support) node.
Master Node	List slave nodes for selected master node.
Slave Node	List constraints for selected slave node.
Member Data	List member data for selected member.
Member Displacements	List displacements for selected member.
Member Forces	List member forces for selected member.
Node Loads	List loads for selected node.

Member Loads	List loads for selected member.
Design Member	List design results for design member containing selected member.
Display Structure Mass	Show mass and location of centre of gravity of the structure..

Note: Query data is displayed in the output window.

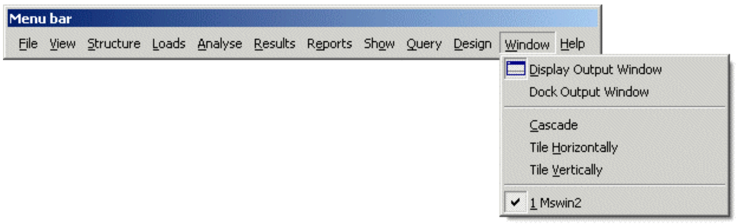
Design Menu Commands



DESIGN MENU

Command	Action
Initialize Design Members	. Sets design code for member. . For RC design, identifies member as beam or column. . Generates default design data for member. . Permits linking of collinear members into a single design member.
Design Data	Input or edit design data. This data includes details of optional intermediate restraints. When design members are initialized they have no intermediate restraints.
Design Load Cases	Select load cases for design of members.
Steel Members	Design or check steel members.
Update Sections	Change sections to those selected in design. For each design group, all designed sections are displayed and you may choose one of these for all members in the group.
Steel Connections	Design or check steel connections.
Reinforced Concrete	Design reinforced concrete members.
RC Detailing	Create CAD DXF of RC details for beams and columns.

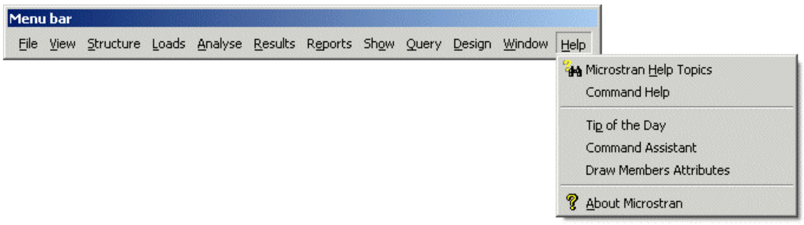
Window Menu Commands



WINDOW MENU

Command	Action
Display Output Window	Show or hide the output window.
Dock Output Window	Repositions output window at bottom of main view.
Cascade	Arranges windows in an overlapped fashion.
Tile Horizontally	Arranges windows above and below.
Tile Vertically	Arranges windows side-by-side.

Help Menu Commands



HELP MENU

Command	Action
Microstran Help Topics	Display Microstran HTML Help. There are three tabs, Contents, Index, and Search, so you can easily find help topics.
Command Help	Show summary of keyboard shortcuts and mouse actions.
Tip of the Day	Show Tip of the Day.
Command Assistant	The Command Assistant offers pop-up instructions for most commands. It may be turned off when no longer required.
Draw Member	Show or hide default attributes for new members.

Attributes	
About Microstran	Display Microstran version and licence details. The Help About dialog box includes links to the internet for support and checking the availability of updates.

Main Toolbar Commands

MAIN TOOLBAR

- Open a new job.
- Open an existing job. Microstran displays the Open dialog box, in which you can locate and open the desired file. This command is for opening an existing job – one for which there is already a Job.msw file, where “Job” is the name of the job as it was saved.
- Save the job with its current name.
- Print the view; i.e. print a picture showing the current view of the structure. Use the **File > Print** command to print a file.
- Print preview; i.e. display exactly how the graphics will be printed. Use the **File > Preview** command to preview a file.
- Standard Structures Input. Create a new job by choosing from one of several types and then inputting parameters. The current job (if any) will be replaced by the new job.
- Table Input.
- Undo the last action. Not available if button is greyed.
- Redo. Not available if button is greyed.

View Toolbar Commands

VIEW TOOLBAR

- Display front view.
- Display right view.
- Display top view.
- Display oblique view.
- Dynamic rotate – drag to rotate.

- Zoom to extents/limits of structure. If the **View > Limit** command is in effect, clicking this button alternately displays the full structure and the limited part of the structure.
- Zoom to window – drag window or click two points.
- Dynamic zoom – drag up or down to zoom, right-click to stop.
- Pan – drag to pan, right-click to stop.
- Limit > Window.
- Limit > Boundary.
- With this button depressed view is limited to set selected in set combo box; otherwise, set is selected.
- With Limit Set button depressed, members in selected set are selected. With Limit Set button not depressed, Limit > View Set command is invoked for selected set.
- Full View – this button reverses effect of Limit command.
- Show/hide the output window.

Display Toolbar Commands

DISPLAY TOOLBAR

- Display symbol at origin of global axis system.
- Display node symbols.
- Display of node numbers.
- Display member numbers.
- Display section numbers.
- Display supports.
- Display pins.
- Display *y* axis for all members.
- Display dialog box showing mass and centre of gravity of structure.
- Display symbols identifying steel connections.
- Display rendered view of structure.
- Display structure in virtual reality window.
- Display annotation of loads.
- Display annotation of member force or displacement diagrams.
- Increase scale for plotting loads, member forces, or displaced shape.

- Decrease scale for plotting loads, member forces, or displaced shape.

Help Toolbar Commands

HELP TOOLBAR

The Help toolbar contains buttons for the **Help Topics** and **Help About Microstran** commands. The Help Topics command starts Microstran HTML Help while Help About Microstran displays a dialog box showing version and licence details. The Help About dialog box includes links to the internet for support and checking the availability of Microstran updates.

Draw Toolbar Commands

DRAW TOOLBAR

- Draw members.
- Erase members.
- Move members.
- Copy members (linear).
- Copy members (polar).
- Reflect members.
- Sub-divide members.
- Intersect members.
- Rotate members.
- Display grid points and set Grid snap mode.
- Set Middle/End snap mode.
- Set Intersection snap mode.

Attributes Toolbar Commands

ATTRIBUTES TOOLBAR

- Input fixed supports.
- Input section numbers.
- Input section properties.
- Input material properties.
- Input member releases.
- Input member orientation reference node/axis.
- Input master-slave constraints.
- Copy attributes from one member to others.

Load Input Toolbar Commands

LOAD INPUT TOOLBAR

- New load case.
- Current input load case – "P" indicates a primary load case and "C" indicates a combination load case. Click the arrow button to display a drop-down list box containing all existing load cases. You may input loads for the displayed load case. When no load case is displayed, load input mode is not in effect.
- Exit load input mode.
- Input node loads – greyed if not inputting a primary load case.
- Input member loads – greyed if current input case is not a primary load case.
- Input combination load case – greyed if current input case is not a combination load case.
- Input area load – greyed if current input case is not a primary load case.

Results Toolbar Commands

RESULTS TOOLBAR

- Display undisplaced structure.
- Select load cases for display.
- Display applied loads. If you find that loads are still displayed when this button is not depressed, load input mode is in effect. Click the red "stop sign" button on the Load Input toolbar to exit load input mode.
- Display member actions. You must turn on this "switch" before you are able to select member forces for display.
- Display axial force, Fx.
- Display shear force, Fy.
- Display shear force, Fz.
- Display torque, Mx.
- Display bending moment, My.
- Display bending moment, Mz.
- Display displaced structure.
- Display reactions.
- Display natural vibration modes.
- Display buckling modes.
- Display design ratios. Design ratios are displayed graphically with different colours representing distinct ranges of values for the percentage of code capacity. For example, members shown bright red are loaded in excess of 110% of the design code capacity.
- Display member force envelope.
- Animate modes (natural or buckling). Each mode is displayed in turn. Press the space bar to move to the next mode or Escape to exit mode animation.

Steel Design Toolbar Commands

STEEL DESIGN TOOLBAR

- Display steel design restraints.
- Input steel design data.
- Copy steel design data.
- Check all steel design members.
- Check selected steel design members.
- Query steel design member.

OK/Cancel Toolbar Commands

OK/CANCEL TOOLBAR

The OK/Cancel toolbar is an alternative to the context menu for confirming or cancelling selections. This toolbar is not displayed initially. Display or hide it with the **View** > **Toolbars** command.

Extra Buttons Toolbar Commands

EXTRA BUTTONS TOOLBAR

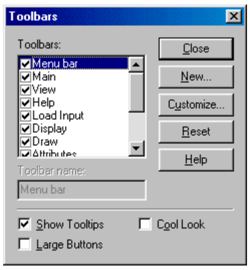
The Extra Buttons toolbar contains a number of buttons that may be added to other toolbars during customization. It is not displayed initially. Display or hide it with the **View > Toolbars** command.

- Redraw (F5).
- Display back view.
- Display left view.
- Rotate viewpoint left. *
- Rotate viewpoint right. *
- Rotate viewpoint up. *
- Rotate viewpoint down. *
- Zoom in.
- Zoom out.
- Insert node.
- Member properties.
- Node properties.
- Create member set.

* May be configured to rotate structure instead of moving viewpoint.

Selecting Which Toolbars Are Displayed

You may easily determine the toolbars that are displayed with the **View > Toolbars** command. This displays the dialog box shown below. All checked toolbars are displayed.



TOOLBARS DIALOG BOX

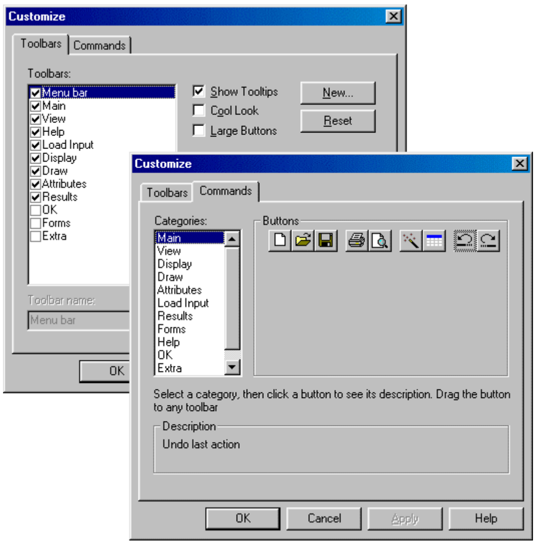
You may also choose the flat style for toolbars or large buttons (these may be preferable at high screen resolutions). Any toolbar that has been customized may be reset to the original configuration by selecting it and then clicking the Reset button.

Customizing Toolbars

As well as being dockable, toolbars in Microstran are customizable in two ways.

Firstly, while pressing the Alt key you may drag any button to any position on the same or another toolbar. If you drag a button to a new position not on a toolbar, it will disappear.

Secondly, you may click the Customize button in the Toolbars dialog box (**View > Toolbars** command). This displays the Customize property sheet. Clicking the New button creates a new empty toolbar with any specified name. On the Commands tab you may now select any existing toolbar and drag its buttons onto the new toolbar (or any other toolbar).



CUSTOMIZING TOOLBARS

The Ouput Window

While the output window is docked you may drag the inner edge to change its size.

The output window, normally at the bottom of the main window, is dockable. You may click on any part of the edge of the output window and drag it so that it floats anywhere on the primary or secondary display or docks on any edge of the main window. You may double-click on the title bar of the floating output window and it will return to its previous docked position. Click the output window button to hide or display the output window.

The position of the output window and each toolbar is persistent from run to run.

Reset Control Bars

Control bars (i.e. toolbars or the output window) may be inadvertently positioned off the visible screen. If you have “lost” any toolbar or the output window you can easily restore program defaults by executing the MsReset.exe program located in the Microstran \Program folder.

Toolbars can also be reset with the **View** > **Toolbars** command.

4:Structural Modelling

General

Structural modelling is the process of representing a structure, and any loads applied to it, as a mathematical model suitable for analysis by the computer.

The structure is represented by a set of *nodes*, each of which usually corresponds to a joint, and a set of *members* connecting the nodes together. In Microstran, each part of the structure is idealized as a line element – line elements are also known as beam, frame, or truss elements. During the structural analysis process, the stiffness properties of the structure are calculated and for each load case the elastic displacements of each node are determined. The resulting distortions of each member are obtained from the displacements of the nodes and then the corresponding member forces are calculated.

Nodes

In a space frame, each node may have up to six degrees of freedom (DOF) – translation in the three global axis directions and rotation about these axes. Each node is free to displace under load in the six global directions unless that node is restrained. The analysis process will compute the displacement component in each of the six global directions where *node restraints* have not been applied. See “Node Coordinates” on page 82 and “Fixed Supports (Node Restraints)” on page 87.

Structure Type

In Microstran, each model has a structure type, *space frame*, *plane frame*, *grillage*, *space truss*, or *plane truss*. For all structure types except the space frame, restraints are automatically applied to all nodes to enforce the displacement pattern implicit in the structure type. For example, a plane frame in the XY plane will have node restraints applied so that displacements out of the plane (Z translation) and rotations in the plane (X and Y rotations) are prevented. See “Structure Type” on page 81.

Members

By default, each member in a framed structure is rigidly connected to the nodes at the ends of the member. *Member releases* or *semi-rigid connections* may be introduced at the ends of the members. When the structure type is either space truss or plane truss member releases are automatically introduced at each end of all members. Axes are associated with each member and these are used as a frame of reference for member loads and member forces. See "Member Definition" on page 83.

Trusses

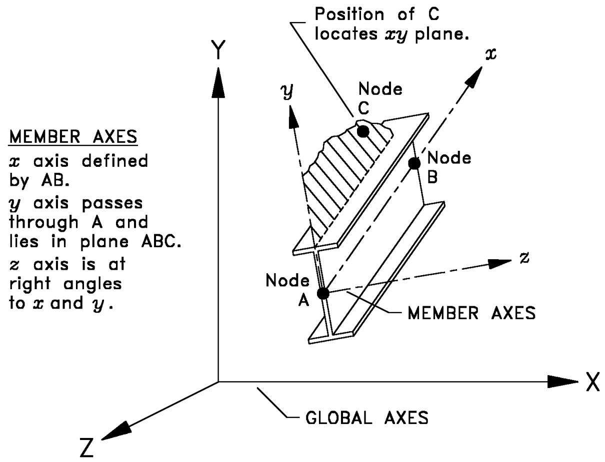
In practice, a "truss" is usually modelled as a frame, not as a truss. The truss structure type is used for pin-jointed models, while the frame structure type must be used if you wish to determine bending moments in *any* member of the structure. For example, if you wish to determine secondary moments in a roof truss the structure type should be plane frame.

Load Cases

There are usually a series of primary load cases each representing separate conditions such as dead load, live load, wind load etc., with load combinations assembled from the primary load cases to represent design conditions for strength and serviceability. Many types of load are available; gravity loads, concentrated node loads, concentrated and distributed member loads, axial and transverse temperature gradients, and member distortions. Each load case comprises any number of these loads in any combination.

Coordinate Systems

Two types of coordinate system are used to describe the structure – the *global axes* and a set of *member axes* for each member. These are shown in the diagram below.



GLOBAL AND MEMBER AXES

Global Axes XYZ

The global axes are a right-handed set of orthogonal axes used to define the geometry of the structure. Node loads and displacements are defined with respect to the global axes. Member loads may also be defined with respect to the global axes. The global axes may be oriented in any direction but usually, either the Y or Z axis is directed upwards.

Member Axes *xyz*

The section axis YY is usually aligned with the y member axis but you may align the XX section axis with y instead. See "Section Properties" on page 88.

Member axes (also referred to as local axes) are sets of right-handed orthogonal axes attached to each member in the structure. The *x* axis runs along the length of the member coincident with the member centroidal axis while the *y* and *z* axes are usually coincident with the principal axes of the cross-section. The positive direction of the *x* axis is determined by the order in which the end nodes of the member are specified. The first node is the "A" node and the second is the "B" node. A third node, the "C" node, is used to define a plane containing the *y* axis. A global axis may be used instead of the "C" node. Member loads may be specified with respect to member axes. Member forces and displacements are reported with respect to the member axes, except in the case of angle sections where you have elected to use principal axis properties.

Angle Sections

With angle sections the rectangular axes and member axes are always located by the reference node/axis but you have the choice of using either *rectangular* axis properties or *principal* axis properties. This choice is made when you select the section from the steel section library. Transverse member loads are always referred to the member *y* and *z* axes (rectangular).

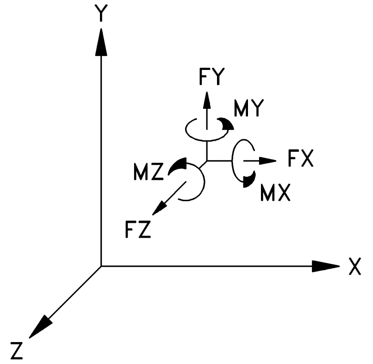
When you choose rectangular axis properties they are used as if they were principal axis properties and results are referred to the member *y* and *z* axes (rectangular). This choice is appropriate when bending is not significant or the section is assumed to be constrained to bend about a rectangular axis.

When principal axis properties are used the member principal axes are rotated from the member *y* and *z* axes and results are referred to these (rotated) axes. Choose principal axis properties for angles that are free to bend about the principal axes.

Sign Conventions

In general, items are positive in the positive direction of the axes to which they refer. For angular displacements and moments, the “right-hand” rule applies, that is, actions are positive if they act in a clockwise sense when viewed along the positive direction of the axis about which they act.

Node loads, displacements and reactions are specified with respect to the global axes. Loads applied to members may be specified with respect to either the global axes or the member axes. The diagram below shows positive node load components.

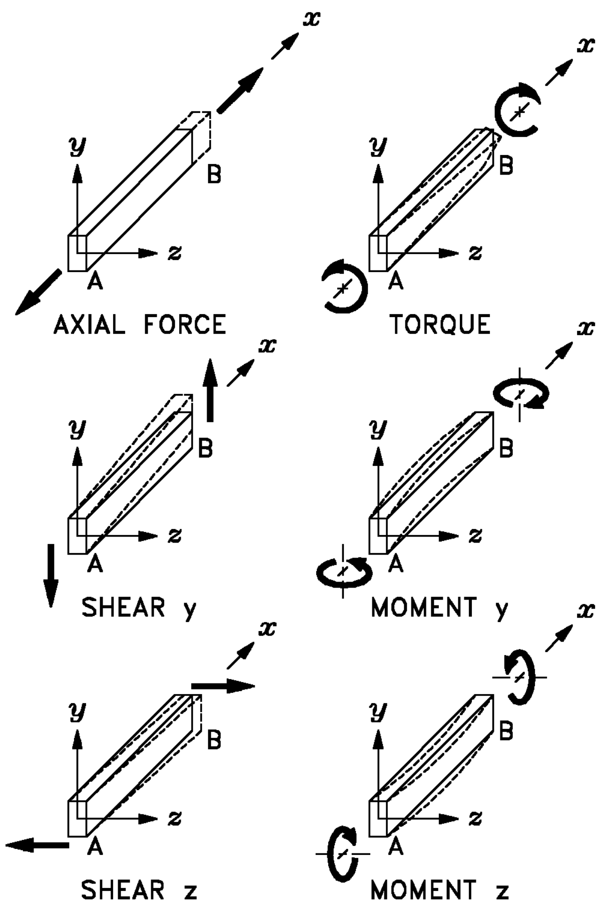


POSITIVE NODE LOAD COMPONENTS

Member end forces and moments are computed with respect to the member axes. An “engineering” sign convention is used so that positive bending moments cause sagging and negative bending moments cause

hogging. Positive member end actions and the corresponding distortions are shown in the diagram below.

Shear force and bending moment diagrams are plotted in the corresponding principal plane of the member. Unless a negative scale factor is set, bending moment diagrams are displayed on the tension side of the member. Axial force and torque diagrams, not being associated with either principal plane of the member, are plotted in the display plane at right angles to the member. The sign convention for plotting axial force and torque depends on the projection of the member on the display plane. If the gradient of the *x* axis is positive, positive values are plotted on the upper side of the member. To assist in interpretation, compression is shown in a broken line.



POSITIVE MEMBER FORCES

Numbering Sequences

Numbers are used throughout Microstran to identify structure elements (such as nodes, members, and section properties) and load cases. You may leave gaps in numbering sequences for maximum convenience in entering or generating data and in interpreting the results. If parts of the structure are added or deleted, there is no need to renumber nodes and members. There is no penalty either in solution time or in storage requirements when gaps occur in any numbering sequence.

Numbering Nodes

The automatic use of the profile optimizer means that you are not constrained in the way you number nodes. The profile optimizer performs internal renumbering of the nodes to minimize the profile of the structure stiffness matrix (see Chapter 14 – "Analysis"). Use the **Structure > Renumber** commands to rearrange node numbering at any stage.

Numbering Members

Solution time and storage requirements are quite independent of the sequence in which member numbers are assigned. It is often convenient to allocate member numbers in blocks for each type of member (e.g. beam, column, chord etc.). Use the **Structure > Renumber** commands to rearrange member numbering at any stage.

Numbering Section Properties

Microstran steel member design programs use the concept of a "design group". All members which have the same section number may be treated as a single design group, allowing the design to automatically take into account the forces and moments in every member in the group. If a design program is being used it is advisable to consider the sequence in which section properties are numbered. Each group of members that must be of the same section should have a unique section number. During the design process, the section used for any design group may be changed.

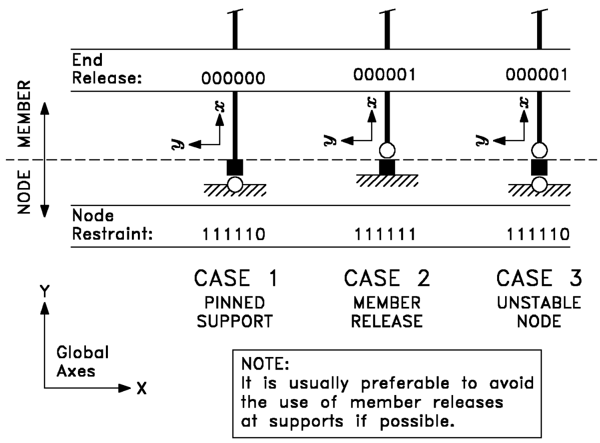
Node Restraints & Member Releases

Member releases are used to create internal hinges in a framed structure and they should not be confused with *node restraints*, which define the supports of the structure. The diagram below illustrates the modelling of a pinned support at the base of a column in a plane frame. The node is shown as a solid block to highlight the difference between a "pin" resulting from the absence of a node restraint and that resulting from the inclusion of a member release.

Case 1 shows the usual situation with a pinned support and no member release. The node restraint code is 111110 (i.e. all DOF restrained except Z rotation) and the member release code is 000000 (i.e. NO releases).

Case 2 shows an alternative arrangement with a fixed support and a member release. The node restraint code is 111111 (i.e. restraint at all DOF) and the member release code is 000001 (i.e. release about member *z* axis only). The analysis would give identical results but the rotation at the base of the column would be unknown because it is not fixed to the node (whose rotation is known).

Case 3 shows a pinned support AND a moment release in the column. This structure is unstable and analysis would stop with a "zero stiffness" or "structure unstable" error message for the support node.



MODELLING A PINNED SUPPORT

For further examples of the use of node restraints and member releases, see "Fixed Supports (Node Restraints)" on page 87 and "Member Definition" on page 83.

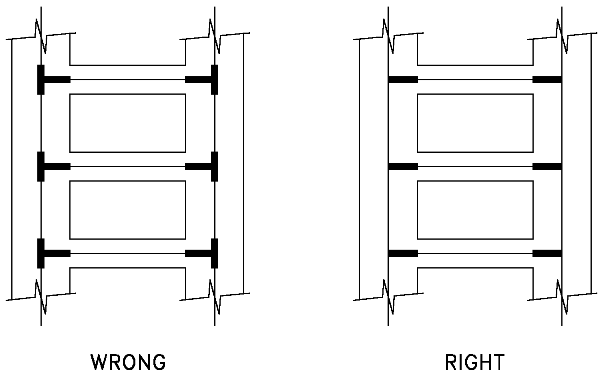
Haunches

All members in a Microstran structural model are prismatic line elements, so that a tapered member must be modelled as a number of segments, each of a different cross-section. Haunches in portal frames can usually be modelled satisfactorily with two such segments. The section properties at the mid-point of the tapered segment may be used for the corresponding prismatic member.

Standard Structures Input automatically computes the section properties of haunches fabricated with specified steel sections. Each haunch may be modelled with as many as 6 segments. See "Single Bay & Multi-bay Portal Frames" on page 142.

Coupled Shear Walls

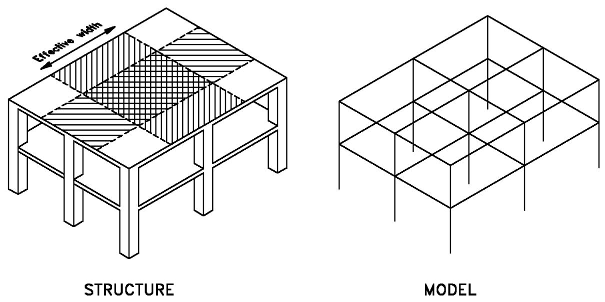
Rigid member offsets prevent all deformation, including axial deformation, between the node and the adjacent end of the member. As a result, rigid offsets should be used with caution in structures where the behaviour depends significantly on the axial deformation of some members. For example, where shear walls or large columns are coupled by beams, if rigid links are used in the column members throughout the depth of the beams, the axial stiffness of the columns would be overestimated. As a result, sidesway of the structure under lateral load would be underestimated. The recommended method of modelling coupled shear walls is illustrated below.



COUPLED SHEAR WALLS

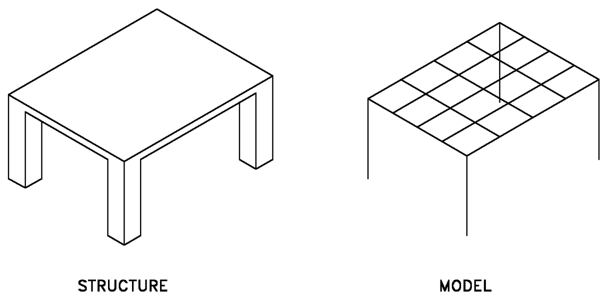
Concrete Slabs

Two-way slab systems are often designed using the “idealized frame” method in which a fairly regular building is assumed to behave as two series of equivalent frames at right angles. For each column grid line there is an equivalent frame in which the properties of each horizontal member are usually assumed to be those of the full width of the slab between the centres of the adjacent spans at right angles.



EQUIVALENT FRAME METHOD

A refinement of this method involves the use of a grillage to model the slab. It is convenient because it leads to a rational distribution of reinforcement when each beam element is uniformly reinforced for the design actions in it. A large model may result for structures with more than a few spans.



GRILLAGE SLAB MODEL

Here are a few recommendations for using this technique:

- A sub-division of each span into about 8 beam strips in each direction would usually provide very accurate results.
- The arrangement of the beam strips may be varied to conform to the shape of the slab; to fit around an opening, for example. The nearest beam strip should be as close as possible to the centroid of an edge beam.

- The properties of each member should be those of the rectangular section of the beam strip with the exception that the following reduced value should be used for the section torsion constant.
  $J = 1/6 \times B \times T^3$
  where B is the width of the beam strip and T is the slab thickness.
- There must be equivalent members in both directions. The total volume of beam strips will be twice the volume of the slab, so if using a GRAV load for calculating self-weight use half the actual density for the slab material.

Multi-storey Concrete Buildings

It is not uncommon to analyse a reinforced or prestressed concrete building as a frame even though it may contain planar elements such as walls and floor slabs.

Reinforced Concrete Service Cores

Generally, a service core may be modelled as an equivalent column located at the centroid or the shear centre of the cross-section. For each horizontal member framing into the equivalent column, a rigid offset is used for the region between the face of the service core and the node of the equivalent column.

Calculation of the torsion constant of the equivalent column presents difficulties if the service core cross-section is capable of warping. For a closed cross-section warping is small and the torsion constant gives a reasonable measure of the torsional stiffness of the core. However, if the service core has an open cross-section or if lintels provide insufficient coupling between adjacent walls for it to be regarded as a closed cross-section, warping will occur in torsion. Such cross-sections are torsionally flexible for uniform torsion (St. Venant torsion) but stiff for non-uniform torsion (which occurs because of the vertical distribution of any torsional load and also because the base of the core is fixed). The torsion constant is a measure of the stiffness of a member in uniform torsion so it underestimates the torsional stiffness of a building core whose cross-section can warp. A possible approach to this problem is to compute the torsion constant as if for a closed section and to reduce it by an empirical factor that takes into account the reduced torsional stiffness of a core that warps.

If you are uncertain about how a service core behaves, it may be advisable to consider two or more alternative models in which different assumptions may be tested.

Concrete Slabs

In analysing the building frame, it is usually necessary to take into account the contribution of the floor slabs in the resistance of horizontal loads. In effect, the slab is replaced by a network of equivalent beams connecting each vertical element to adjacent vertical elements. Care should be taken in considering what values to use for the effective width

of these beams. Generally, the full width of the slab, used in the design of the slab itself, is not used (see “Concrete Slabs” on page 69). It may be reasonable, for instance, to use code rules regarding the effective width of a tee beam.

Usually, a concrete floor slab can be regarded as being rigid in its own plane (i.e. in plan, the floor undergoes rigid-body motion only). If the Microstran Master-Slave Constraints feature is available, this fact can be used to permit the saving of three DOF at each node in the floor except for one arbitrarily selected as the “master” node (see “Master-Slave Constraints” on page 96). The accuracy of the solution may also be improved by the inclusion of master-slave constraints. The network of equivalent beams should be arranged so that “racking” of the floor is inhibited if master-slave constraints are not used.

Inputting a Multi-storey Model

A systematic procedure should be adopted for inputting a large structural model. The procedure set out below, which uses Graphics Input, has been used successfully for multi-storey models. It is a good idea to use the **File > Save As** command to save the job with a new name at critical stages.

a. Create typical floor module

Start by drawing equivalent beams in the XY plane. Graphics Input is very convenient because the model can be input directly from the typical floor plan. Draw beams from left to right and from bottom to top on the screen. It does not matter how the software allocates node and member numbers because these can easily be rearranged in order. Note that member incidence (the “direction” of the member) is not changed by the sorting operation. When the floor is finished, copy all members to the level above with *node extrusion*. This creates a column at each node – erase any that are not required. Erase all beams at the lower level. Input all sections and materials, using arbitrary values at this stage if necessary.

b. Sort nodes and members

Use the **Structure > Renumber** command to sort node and member numbers. The primary and secondary sort directions should be Z and Y respectively. The “Connect collinear members” option may be selected if desired.

c. Input rigid offsets and master-slave constraints

Rigid offsets and master-slave constraints are most conveniently input using Graphics Input.

d. Propagate typical floor module

Change the vertical axis from Y to Z using the **Structure > Type** command. Copy the typical floor module to a new job and in the new job, use the **Structure > Copy > Linear** command without extrusion to produce as many copies of the typical floor module as required.

e. Add non-typical parts

Any parts of the structure that were not generated by the propagation of the typical floor should now be added.

f. Input restraints, loads etc.

Any convenient method may be used to complete the model.

The model representing the typical floor module should be maintained as a separate job because the procedure of building up the model will have to be repeated if changes are required.

Instability & Ill-Conditioning

Instability occurs if a structure or a part of it can move without any resistance. In this case no solution can be obtained. *Ill*-conditioning occurs if a solution is obtained but with severe loss of numerical precision. An ill-conditioned structure is usually one that is almost unstable, producing gross displacements during analysis. Ill-conditioning may also occur if there is an excessive difference in stiffness from one part of the structural model to another.

Instability and ill-conditioning are usually caused by insufficient node restraints, too many member releases, or inappropriate values for some section properties. With non-linear analysis, buckled members may be removed automatically from the model, thereby causing instability. Elastic critical load analysis is an invaluable tool for detecting problems of this type.

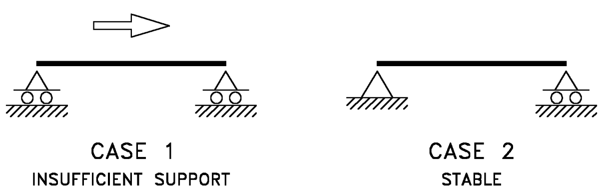
The most common error conditions are described in this section. See also "Common Modelling Problems" on page 75.

Insufficient Support

If insufficient supports have been specified for overall stability of the structure, analysis is not possible. This condition will be detected before analysis is started and an error of the following form will be reported:

`STRUCTURE HAS INSUFFICIENT SUPPORT`

Case 1 in the diagram below shows a simply supported beam with insufficient support.



INSUFFICIENT SUPPORT

Zero Stiffness at Node

If there is zero stiffness for any DOF at any node a force or moment could be applied at that DOF without any resistance and so it would be impossible to calculate a displacement in that direction. These errors are detected at an early stage of analysis and reported with an error of the form:

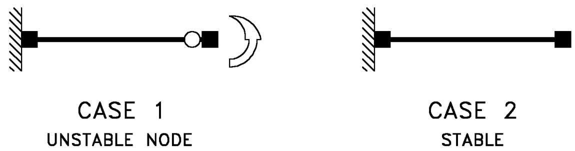
```
ZERO STIFFNESS AT NODE nnnnn DOF f
```

where:

nnnnn	=	The node number at which instability was detected.
f	=	The DOF number, as shown in the table below, in which there was found to be no resistance to displacement.

DOF	Direction	DOF	About Axis
1	X	4	X
2	Y	5	Y
3	Z	6	Z

A node where a zero stiffness error occurs may be referred to as an unstable node. Unstable nodes can be eliminated by applying a restraint to the zero stiffness DOF at the node. An unstable node is shown in Case 1 in the diagram below. Nodes are represented as solid blocks.



ZERO STIFFNESS AT NODE

Mechanism Instability

If a structure has any part that can move freely, it is actually a mechanism and cannot be analysed. A mechanism will be detected during analysis as an attempt to perform impossible numerical operations at a particular DOF and one of the following errors will be reported:

```
STRUCTURE UNSTABLE AT NODE nnnnn DOF f
```

or

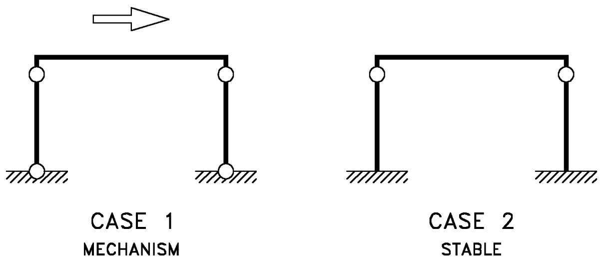
```
ZERO PIVOT AT NODE nnnnn DOF f
```

where "nnnnn" and "f" are as defined above.

Mechanisms may be eliminated by applying a restraint to the DOF reported in the error message but you should consider whether the error is a symptom of a more fundamental problem with the structural model

such as a missing support, an inappropriate section property value, or an incorrect member release.

A mechanism is shown in Case 1, in the diagram below, where the frame has pinned supports and also has pins at the top of each column. If any one of the four pins were removed, the structure would be stable.



MECHANISM INSTABILITY

Zero stiffness at a node in a direction that is not parallel to a global axis is also detected as a mechanism during analysis. This type of instability cannot be eliminated by specifying a restraint (in a global axis direction) but by adding a link member or a spring in a non-global axis direction.

Ill-Conditioning

If severe loss of numerical precision occurs during analysis the structure is ill-conditioned. The condition number, reported on the screen after analysis, is a measure of how much precision was lost. When an analysis has been completed with a large condition number being reported, the results may be meaningless and the structure may well be unstable. A displaced shape plot of an ill-conditioned structure would usually show gross displacements. Microstran cannot determine at what point the ill-conditioning becomes unacceptable so that you should always check the condition number (see “Report Contents”). A warning is generated if the condition number exceeds $10E^4$.

After analysis Microstran performs an equilibrium check in which the external forces are compared to the sum of the member forces at each node. If this check shows a discrepancy the following error message will be reported:

```
LARGE RESIDUAL - POSSIBLE MODELLING ERROR
```

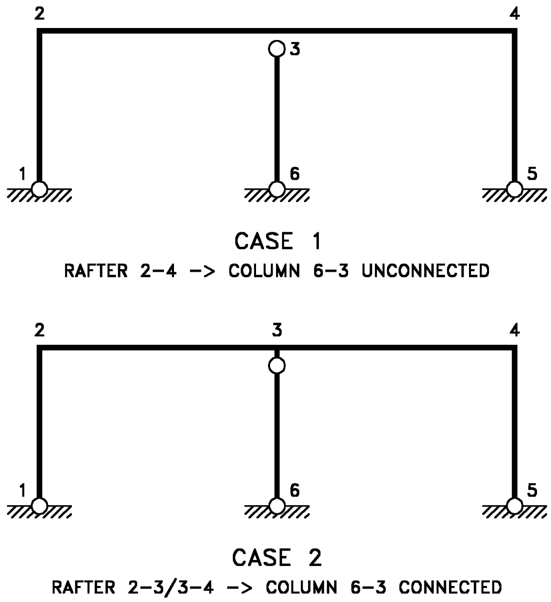
In this case, a large condition number will also be reported and the results should be regarded as being meaningless.

Important Note: It is possible for ill-conditioning to occur without a discrepancy in the equilibrium check – a satisfactory equilibrium check is not a guarantee that the analysis was successful.

Common Modelling Problems

Unconnected Members

Sometimes, it may not be evident that a single member has been input instead of two or more connected end-to-end. If this occurs, it can give rise to unconnected members at the interior nodes. In the diagram below, for example, if the rafter is input as a single member (2-4) from node 2 to node 4, the central column will not be connected to the rafter and it will be unstable. The rafter must be input as two members, 2-3 and 3-4, so that the central column (6-3) is connected to it.



UNCONNECTED MEMBER

Unconnected members are detected in the checking phase that precedes analysis. You may use the **Analyse > Check Input** command to see warnings about unconnected members. The warning message displayed in the output window for the structure in Case 1, above, would be similar to:

```
Nodes adjacent but not connected to members:
Warning: node 3, member 2
```

Plane truss and space truss structure types are rarely used nowadays. Using these structure types saves RAM but makes analysis more difficult for the user.

Wrong Choice of Structure Type

Structures that are referred to as *trusses* are usually modelled as plane or space frames with member releases introduced as required. When the structure type (as determined by the **Structure > Type** command) is *plane truss* or *space truss*, Microstran globally eliminates bending actions from the member stiffness formulation and every member behaves as if it is *pin-ended*. The structure type must not be "plane truss" or "space truss" if any of the following conditions exists:

- You want to determine bending moment at any point in the structure.
- Stability of the structure depends on bending moment at any point.
- The structure is not fully triangulated.

If you attempt to analyse a structure that is incorrectly classified in Microstran as a truss, a "zero pivot" error will result – see "Mechanism Instability" on page 73.

Angle Between AB and AC too Small

```
Member nn ref. node/axis –
 angle between AB and AC too small
```

This error message may be displayed in the output window during the checking phase prior to an analysis. It occurs if the specified reference axis for a column is the vertical axis and in any other situation where the vector to the reference node is almost parallel to the longitudinal member axis (x).

To understand what the error message means, refer to the diagram showing global and member axes in "Coordinate Systems" on page 63. The member x axis is AB and the y axis is computed using the vector cross-product of AB and AC. When the angle between AB and AC is zero the computation is impossible. If the angle is very small the computation is uncertain.

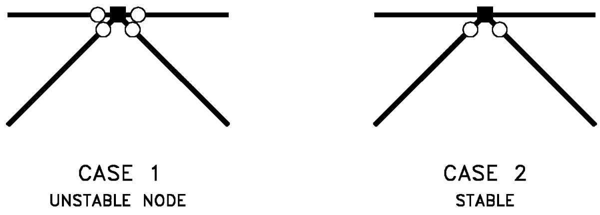
Fix the problem by double-clicking on the affected member and changing the reference node/axis. In many cases, the default reference axis is suitable.

Too Many Releases

If a zero stiffness error is reported at a node in a plane frame or space frame it is usually because all members connected to the node are pin-ended. This situation, illustrated in Case 1 in the diagram below, may be avoided by any of the following procedures:

- Set the structure type to plane truss or space truss (as appropriate), in which case Microstran will automatically treat all members as pin-ended and delete rotational DOF.
- Apply restraints at all DOF where zero stiffness is reported.

- In a truss, remove pins in the chord members (often continuous) leaving the web members pin-ended (see Case 2, below).
- Remove all pins. If the structure is fully triangulated, bending moments will usually be insignificant but if they are not, releases may be inserted as required. This is sometimes the best approach for a preliminary analysis.



CONTINUOUS CHORD IN PLANE TRUSS

Coplanar Nodes in Space Trusses

In a space truss (where all members are pinned at both ends), *the structure must be fully triangulated to be stable*. Any node in a space truss where all connected members lie in a plane is known as a *coplanar* node and will be unstable unless there is a restraint, a spring support, or a master-slave constraint that inhibits displacement normal to the plane. In the Microstran checking phase, warning messages are issued for all coplanar nodes detected.

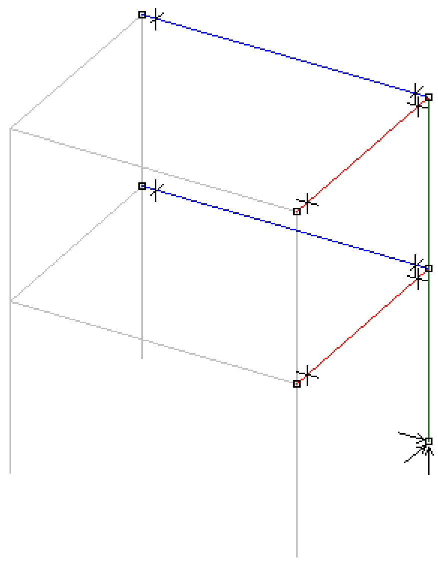
Consider, for example, a tower structure modelled as a space truss (see "Example 3 – Space Truss" on page 349). Each face panel of the tower is a plane truss and although it may be stable in its own plane, out-of-plane displacement of the interior nodes of the truss must be inhibited to make the model stable. If the analysis proceeds with any instability of this type existing, a zero stiffness error will occur if the plane is parallel to a global plane. If it is not, instability or ill-conditioning will be reported during analysis.

Large Differences in Stiffness

When very stiff structural elements are being modelled, unreasonably large values should not be input for section properties. For example, when modelling a wide column, a very stiff link member may be used to connect a node on the centre-line with a node on the face of the column. While a value for the moment of inertia of the link of $10E^3$ times the moment of inertia of the column may yield satisfactory results, a value of $10E^{20}$ times that moment of inertia would cause ill-conditioning. This kind of modelling problem can be eliminated altogether by the use of rigid member offsets (see "Rigid Member Offsets" on page 94).

Columns Without Rotational Restraint

In a space frame where all beams connected to a column have moment releases about a vertical axis, there will be a problem if the node at the base of the column does not have a rotational restraint about the vertical axis. This condition is illustrated in the model shown below.



COLUMN WITHOUT ROTATIONAL RESTRAINT

Usually, analysis will fail with a mechanism instability corresponding to rotation of the column about its axis. If the nodes of the column do not lie exactly on a vertical line (e.g. the coordinates of one of the column nodes are slightly in error), Microstran may not detect a mechanism instability. In such a case, there would be an excessively high condition number and there would be excessive rotation of the column nodes about the vertical axis.

Note: A beam with all connected members pinned creates a similar situation.

5:Structure & Load Data

Input Methods

Microstran offers a variety of input methods that may be used interchangeably in any job. The data required is described in this chapter and the different input methods are discussed later.

Graphics Input is like a dedicated CAD system for entering structure and load data. This input method is easy to learn and offers the greatest productivity in most situations. It is very useful when making changes, regardless of the original input method.

Table Input is a spreadsheet style of input that provides an excellent overview of the job in a *tree view*. Each data entity type is represented as a branch in the tree and you may expand any branch to show all the entities. Double-clicking on any entity displays a table of all such entities. You may copy to and paste from the Windows clipboard, so it is ideal for interfacing to *spreadsheet programs*, such as Microsoft Excel.

Standard Structures Input provides the quickest way to input many common types of structure. Many configurations of portal frame may be input with the entry of a small number of parameters.

With **Macro Language Input** the structure and load data are described in a text file. Powerful generation facilities make it an ideal method of input for many types of structure. The MLI format is similar to input formats used by some other structural programs including STRESS and STRUDL.

The *archive file* is a text file that contains all structure, load, and design data for the job. **Archive File Input** is the process of importing a text file of this type into Microstran. You may edit any job data by exporting the archive file, editing it with Microstran's text editor, and then importing it back into Microstran. The archive file format also provides the most convenient interface to Microstran for third-party programs.

CAD Input transfers structure data directly from any 3-D CAD system that uses the standard format AutoCAD DXF. Structure data may also be transferred from Microstran to a CAD system.

The **File > Import > Other** command permits the input of data from some other program.

Numbering Sequences

Numbers are used throughout Microstran to identify structure elements such as nodes and members. These numbers are referred to as *entity labels* and they can be any integer from 1 to 99000. If parts of the structure are added or deleted, there is no need to renumber nodes and members. There is no time or storage penalty when gaps occur in a numbering sequence.

Job Title

Two lines of descriptive information may be entered for the job. Up to 60 characters of text are allowed in each line. These lines, which may contain any information desired, appear on all output reports for identification of the job. The job title may be entered with the **Structure > Title** command.

Job Notes

Microstran associates descriptive information with each job. Notes are saved with the job data and are particularly useful for recording the development history of the job. Use the **Structure > Job Notes** command to display notes in a window.

With the Job Notes option selected in General Configuration the notes will be displayed when any saved job is opened.

Units

The units of length, force, mass, and temperature used in a Microstran job may be determined by the user with the **Structure > Units** command. Any data extracted from section or material libraries will be converted automatically to the current units.

On selecting the **Structure > Units** command a set of units may be chosen from the table below. By default, Microstran uses units of meters, kilonewtons, tonnes, and °C.

No.	Length	Force	Mass	Temp.
1	meter (m)	kilonewton (kN)	tonne (t)	°C
2	meter (m)	newton (N)	kilogram (kg)	°C
3	millimeter (mm)	newton (N)	tonne (t)	°C
4	foot (ft)	kip (k)	kip/g (32.2)	°F
5	inch (in)	kip (k)	kip/g (386.0)	°F

Units are automatically converted if you select a set of units differing from that currently in effect.

Note: All Microstran steel design options require the use of the meter and kilonewton units.

Structure Type

Microstran may be used to analyse plane trusses, grillages, plane frames, space trusses, and space frames. When you specify the structure type, Microstran will automatically apply the restraints necessary to enforce the restricted displacement pattern of the specified structure type. Additional restraints must be specified at support nodes.

When a structure type other than space frame is selected, any loads or elastic restraints applied to suppressed degrees of freedom will be ignored. In a plane or space truss all members are considered to have pinned ends. If it is necessary for any member in a truss to be continuous (e.g. chord members), the structure must be described as a frame, not a truss.

Note: In plane truss and plane frame structures member *y* axes and loads must be in the plane of the structure. Grillages cannot have loads in the plane of the structure.

No.	Type	Vert.	X	Y	Z	RX	RY	RZ
1	Plane truss in XY plane	Y	✓	✓	×	×	×	×
2	Grillage in XY plane	Z	×	×	✓	✓	✓	×
3	Plane frame in XY plane	Y	✓	✓	×	×	×	✓
4	Space truss	Any	✓	✓	✓	×	×	×
5	Space frame	Any	✓	✓	✓	✓	✓	✓

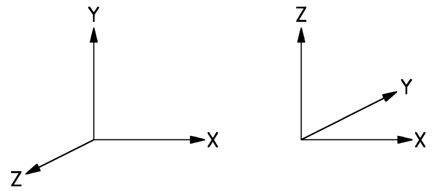
Restraints are applied to translations and rotations of the nodes with respect to the global axes. Suppressed degrees of freedom (DOF) are shown in the table as ×, while remaining DOF are shown as ✓.

Microstran automatically reclassifies plane structures as space structures if any node coordinate is out of plane.

Structure type may be set with the **Structure > Type** command.

Node Coordinates

Node coordinates are defined with respect to a right-handed Cartesian (rectangular) axis system, referred to as the global coordinate system or the *global axes*. Two commonly used orientations of the global axes are shown in the diagram below. Cylindrical and spherical axis systems may also be used in some input modules. Coordinates are automatically transformed by Microstran to global (X,Y,Z) coordinates on input. See "Coordinate Systems" on page 63.



GLOBAL COORDINATE SYSTEM

Rectangular Coordinates

For each node whose coordinates are specified in the global system, the following values are required:

X	Global X coordinate of node.
Y	Global Y coordinate of node.
Z	Global Z coordinate of node.

Cylindrical Coordinates

For each node whose coordinates are specified in cylindrical coordinates, the following values are required:

R	The radial coordinate of the node.
Θ	The angular coordinate measured in degrees about the Z axis, in a right-handed sense from the X axis.
Z	Global Z coordinate of node.

Spherical Coordinates

For each node whose coordinates are specified in spherical coordinates, the following values are required:

R	The radial coordinate of the node.
Θ	The angular coordinate (longitude) measured in degrees about the Z axis, in a right-handed sense from the X axis.
Φ	The angular coordinate (latitude) measured in degrees above the XY plane.

Member Definition

Key concept.

As well as using toolbar buttons or the commands on the **Structure > Attributes** sub-menu to change any property of a member, you may double-click on the member and change any of the values in the Member Property dialog box that is displayed. You can change properties of any number of members by selecting them, right-clicking, and choosing Properties on the context menu. Any value entered applies to all members selected.

Member Connectivity

A	Node "A" – start of member.
B	Node "B" – end of member.

Member connectivity, also referred to as member incidence, is simply the specification of the two nodes joined by the member.

Each member has a set of axes, *x*, *y,* and *z*, referred to as *member* or *local* axes, in which the *x* axis is directed along the centroidal axis of the member from node "A" to node "B". Except in the case of angles, the *y* and *z* axes coincide with the principal axes of the member cross-section at "A".

Section properties, some load types, and computed member forces are specified with respect to the member axes. As a result, data input and interpretation of output are usually simplified if the member axes are oriented in a consistent manner, such as having the *x* axis generally directed either upwards or to the right (as normally viewed). In general, this will be achieved by using the default reference axis. See "Coordinate Systems" on page 63.

Reference Node/Axis

C	Node/Axis "C" – reference node or axis.

The reference node or axis defines the orientation of the transverse member axes, *y* and *z*. See "Coordinate Systems" on page 63.

Together with the longitudinal axis of the member, *x*, the reference node or axis defines a plane that contains the *y* axis. Except in the case of angle sections, the principal axes of the cross-section coincide with the member *y* and *z* axes. As shown in the diagram, the *xy* plane usually coincides with the web plane of a member having an I-shaped cross-section.

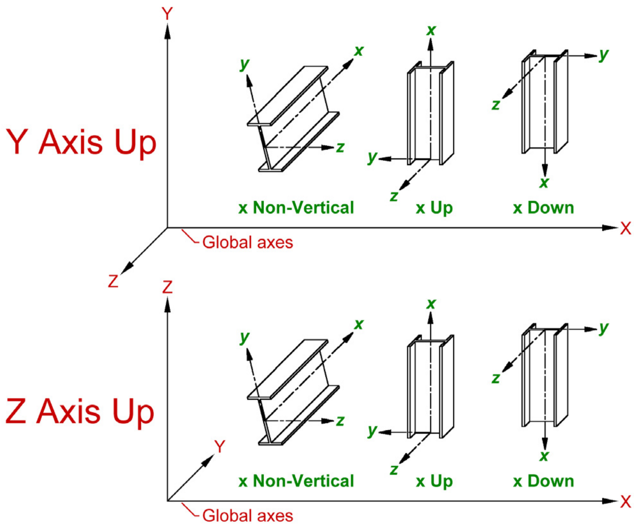
Reference Node
Any node may be used for this purpose provided that it is not collinear with the member (i.e. provided it does not lie on the member *x* axis). If the node is used solely as a reference node it will be restrained automatically by Microstran.

Reference Axis
For members with the member *xy* plane parallel to one of the global axes, as commonly occurs in most structures, it is convenient to use a reference node at an extreme distance along that axis. (Because the dimensions of the structure subtend an infinitesimally small angle at this distant reference node, it may be used for all members with the same orientation.) Thus, Microstran recognizes any of the six global axis directions (X, Y, Z, -X, -Y, -Z) as a reference axis. If a member is parallel to a global axis, that global axis must not be used as a reference axis for the member.

Default Reference Axis
If you leave any reference node/axis unspecified or input a “D”, a value will be assigned automatically. This value depends on the vertical axis of the structure, whether the member is vertical, and if so, whether its longitudinal axis, *x*, is up or down. The default reference axis is set out in the table below for all relevant combinations of vertical axis and member *x* axis direction.

Vert. Axis	*x* Non Vert.	*x* Up	*x* Down
X	X	-Y	Y
Y	Y	-X	X
Z	Z	-X	X

Each non-vertical member will have its *y* axis inclined upwards and each vertical member will have its *y* axis parallel to the horizontal global axis shown in the table. The default orientation of the member axes is shown for typical members in the diagram below.



DEFAULT ORIENTATION OF MEMBER AXES

Input Reference Node/Axis

Use the **Structure > Attributes > Reference Node/Axis** command to set or modify the reference node/axis for one or more members.

Section Number

Each member must be assigned a section number that appears in the list of sections in the input data for the structure – see “Section Properties” on page 88.

Input Section No.

Use the **Structure > Attributes > Section Number** command to input the section number for one or more members.

Material Number

Each member must be assigned a material number that appears in the list of materials in the input data for the structure – see “Material Properties” on page 92.

Member Release Codes

Releases (e.g. pins) are used in framed structures to isolate a member from an adjacent node for one or more components of member force. Released member force components are zero. By default, each member in a framed structure has no releases and is connected rigidly to the adjacent nodes. Member releases must not be input when the structure type is *plane truss* or *space truss*.

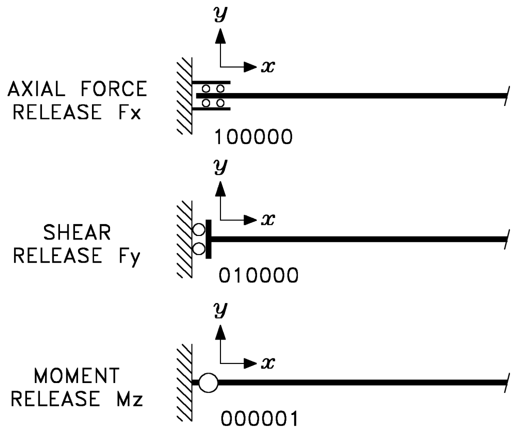
The member release code consists of a string of six digits corresponding to the force components F*x*, F*y*, F*z*, M*x*, M*y*, and M*z*, which are defined with respect to the member axes. A “1” means that a release DOES exist while a “0” means that is does NOT exist. The default member release code is therefore “000000” at each end.

Any combination of member releases may be used provided that the member remains stable and is not completely released from the structure at either end.

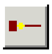
Examples of unacceptable combinations are shown in the table below.

Release Codes	Problem
100000 100000	An axial release at each end would give rise to a member with no resistance to an axial load applied to the member.
000100 000100	A torque release at each end would give rise to a member with no resistance to a torque applied to the member.
111111 000000	A code of 111111 at either end would result in the member being completely isolated from the adjacent node.

Some member releases that occur in plane frames are shown below.



MEMBER RELEASES

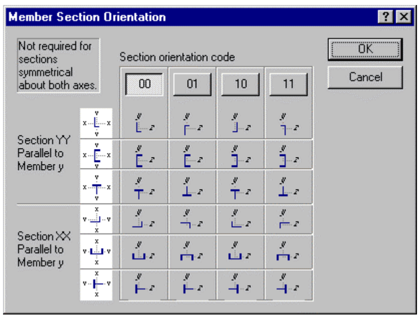


Input Releases

Use the **Structure > Attributes > Member Releases** command to input member releases for one or more members.

Cross-Section Orientation Code

For members whose cross-sections are not symmetrical about both principal axes, it is necessary to specify additional information so that Microstran can show the member with the correct cross-section orientation. This is important for representing the structure accurately in rendered views, and also for the export of information to detailing programs. In the dialog box shown below, the default orientation of the library sections for which this code is applicable is shown in the leftmost column. When the first digit of the direction code is 1 it denotes rotation of the cross-section about the member *y* axis and when the second digit is 1 it denotes rotation of the cross-section about the member *z* axis.



CROSS-SECTION DIRECTION CODE

Fixed Supports (Node Restraints)

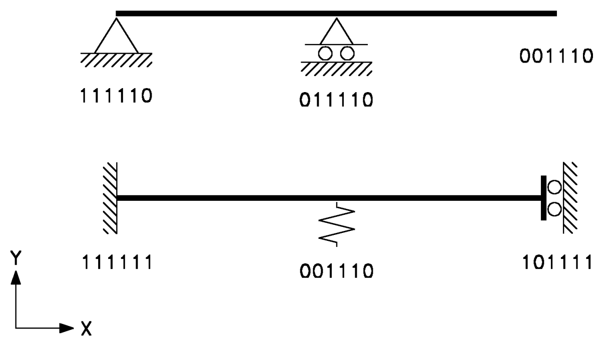
A fixed support is created at a node when restraints are specified in one or more directions at the node. Restraints are specified by the *node restraint code*, a string of six digits. A “1” means that a restraint DOES exist while a “0” means that it does NOT exist. Restraints are specified with respect to the global axes as shown in the following table.

1	2	3	4	5	6
X	Y	Z	RX	RY	RZ
Translations			Rotations		

Fixed support nodes are identified graphically with an arrow for translational restraints and a crossbar on the arrow stem for rotational restraints.

Reactions at each restraint are listed in the reaction table in the analysis report. Loads applied directly at restraints are included in the calculation of reactions. A restraint should be specified in any direction for which a displacement is prescribed in the load data.

Some node restraint codes that occur in plane frame structures are shown below.



PLANE FRAME NODE RESTRAINTS

Input Supports

Use the **Structure > Attributes > Fixed Supports** command to input fixed supports at one or more nodes.

Spring Supports

Spring supports are used to model external elastic supports provided by the foundation or other structures. They are defined by specifying up to six spring stiffnesses at each node where there is an elastic connection between the structure and a point of rigid support. All spring stiffnesses act in the global axis directions and must be specified in consistent units. Spring supports may be applied to any DOF that has not been suppressed because of the structure type. If a fixed support *and* a non-zero spring stiffness are specified at any DOF, the spring support will take precedence. At each node where one or more spring supports exist, the following data is required:

KX	X translation spring stiffness.
KY	Y translation spring stiffness.
KZ	Z translation spring stiffness.
KRX	X rotation spring stiffness.
KRY	Y rotation spring stiffness.
KRZ	Z rotation spring stiffness.

Translational spring stiffnesses are defined in units of force per unit displacement. Rotational spring stiffnesses are defined in units of moment per radian (i.e. the moment required to produce an elastic rotation of one radian). All stiffness values must be zero or positive – a zero value indicates that there is no spring in the corresponding direction.

Nodes with spring supports are identified graphically by a spring symbol.

Use the **Structure > Attributes > Spring Supports** command to input spring supports at one or more nodes.

Section Properties

For each job there is a table of sections, which is referenced by the section number specified for each member. Sections may be input by any of the following methods:

- **Library:** Specifying the name of the section so that the properties may be extracted from a library.
- **Shape:** Specifying the shape and the dimensions necessary for Microstran to compute the section properties. For example, for a rectangular section, the depth and breadth are input.
- **Values:** Entering directly the geometric properties of the section. With this method you may create a custom shape using the

Microstran shape builder, or select an I section and a channel from the library to form a crane beam.

Microstran design options use the concept of a “design group”. This permits all members with the same section number to be designed together. If you wish to use this feature, you should allocate a different section number for each group of members having the same section.

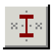
The following items of data may be input for each section, whichever method is used:

Section Name

The section name is used to identify the section when extracting properties from a section library and also in reports. Up to 15 characters are allowed for the section name. Blanks are not permitted.

Section Comment

The comment is used to identify the section in reports. Up to 20 characters may be used.

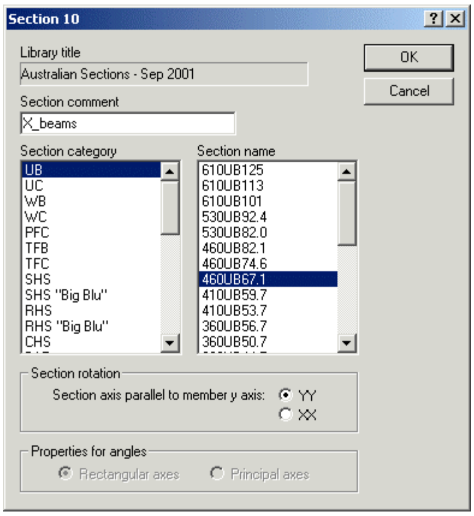


Input Section Properties

Use the **Structure > Attributes > Section Properties** command to choose a section number and input the properties for it.

Selection from Library

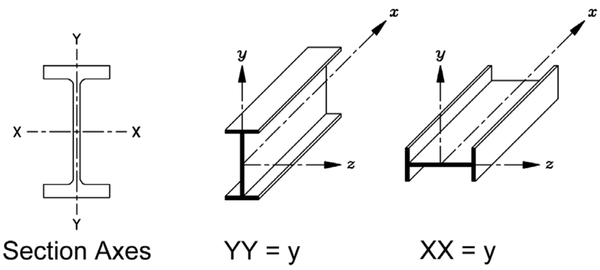
To select a section from the configured library you only need to choose the section category and then the desired section. Standard naming conventions are used in Microstran section libraries. You may edit libraries or make a new one using the **File > Configure > Section Library Manager** command (see Chapter 18 – “Section & Material Libraries”).



SECTION FROM LIBRARY

Section Axis Alignment

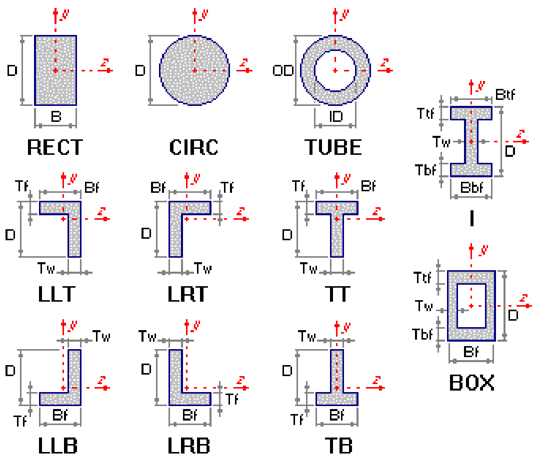
For library sections you may specify the cross-section axis (XX or YY) that is aligned with the member *y* axis (see “Coordinate Systems” on page 63).



SECTION AXIS ALIGNED WITH MEMBER y AXIS

Shape Input

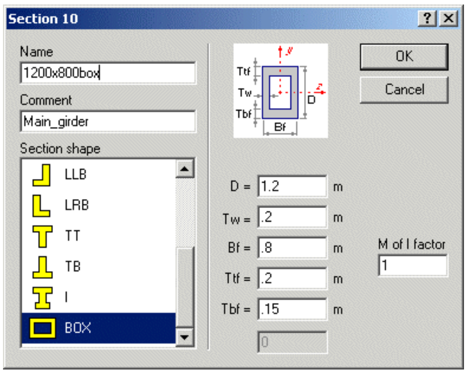
Often the most convenient input method for a section is to specify the shape and corresponding dimensions. The shapes available are shown in the following diagram. In the diagram, the member *y* axis is directed upwards in each case.



SECTIONS AVAILABLE FOR SHAPE INPUT

When entering section data by shape you may specify a moment of inertia factor. The normally computed moment of inertia is multiplied by this factor. It is typically used in reinforced concrete members to take account of reduction in stiffness caused by cracking.

Consistent units must be used for section dimensions (e.g. m where m are used for node coordinates).



SHAPE INPUT

Property Value Input

When property value input is selected, the geometric properties of each member cross-section are be specified numerically. The following data is required:

Cross-Sectional Area, A*x*

The area must be in units consistent with those used elsewhere in the input (e.g. m^2 where m are used for node coordinates).

Shear Area, A*y*

The area that is effective in resisting shear distortion in the member *y* direction. The shear area is required only if it is necessary to account for shear distortion. It must be in units consistent with those used elsewhere in the input (e.g. m^2 where m are used for node coordinates). Shear distortions are usually a very small proportion of the total distortion of any member and are generally ignored in manual calculations. If you enter a value of zero, shear distortions will be ignored in the corresponding members. For I sections loaded in their strong direction the shear area is approximately equal to the area of the web. For solid rectangular sections it is equal to 5/6 of the cross-sectional area. Shear areas stored in section libraries are usually zero (i.e., shear distortion is ignored), however, you may change them by editing the library.

Shear Area, A*z*

The area that is effective in resisting shear distortion in the member *z* direction. See above.

St. Venant Torsion Constant, J

The St. Venant torsion constant must be in units consistent with those used elsewhere in the input (e.g. m^4 where m are used for node coordinates). A value of zero may be input if the members with this section are not required to withstand torque. In this case, you should ensure that members with this section do not also have a torque release at either end.

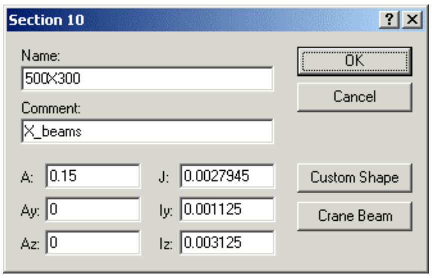
Second Moment of Area, I*y*

The second moment of area (moment of inertia) that is effective in resisting flexural distortion about the member *y* axis. It must be in units consistent with those used elsewhere in the input (e.g. m^4 where m are used for node coordinates

Second Moment of Area, I*z*

The second moment of area (moment of inertia) that is effective in resisting flexural distortion about the member *z* axis. See above.

The dialog box for entry of section property values contains Custom Shape and Crane Beam buttons. The Custom Shape button starts the Microstran shape builder program, allowing you to describe a section graphically. The Crane Beam button displays a dialog box in which you can input a crane beam by choosing an I section and channel from the library.



NUMERICAL VALUES

Material Properties

For each job there is a table of materials, which is referenced by the material number specified for each member. Materials may be input by specifying the name of the material, so that the properties may be extracted from a library, or by entering directly the properties of the material.

The material name may be input for each material, whichever method is used.

Material Name

The material name is used to identify the material when extracting properties from a material library and also in reports. Up to 15 characters are allowed for the material name. Blanks are not permitted.

Input Material Properties

Use the **Structure > Attributes > Material Properties** command to choose a material number and input the properties for it.

Selection from Library

To select a material from a library you only need to specify the name of the library and the name of the material. Microstran is configured with a default material library name so that you will have to enter this only if you are not using the normal library. The material library may be edited as required (see “The Material Library” on page 315).

Property Value Input

When property value input is selected, the properties of each material are specified numerically. The following data is required:

Young’s Modulus, E

Young’s modulus must be in units consistent with those used elsewhere in the input (e.g. kN/m^2 – kilopascals – where m and kN are used).

Poisson’s Ratio, μ

Poisson’s ratio is used to compute the shear modulus from the expression:

$G = 0.5 \times E / (1 + \mu)$

Typical values are 0.25 for steel and 0.15 or 0.20 for concrete.

Mass Density

The mass density is required for the calculation of acceleration loadings and the structure mass. The unit used for mass must be consistent with the force, length, and time units: If m and kN are being used, material mass densities should be in t/m^3. Typical values for steel and concrete are 7.85 t/m^3 and 2.5 t/m^3, respectively.

Coefficient of Thermal Expansion

The coefficient of thermal expansion is only required if thermal loads are applied to the structure. A value of $11.7E^{-6}$ per °C is typically used for both steel and concrete.

Node Mass

In dynamic analysis, the mass of each member is automatically computed and assigned to the nodes at the ends of the member. Mass additional to that of the frame may be input at any node. The units must be consistent with those used elsewhere in the input, for example, where meters and kilonewtons are used, the unit of mass will be the tonne.

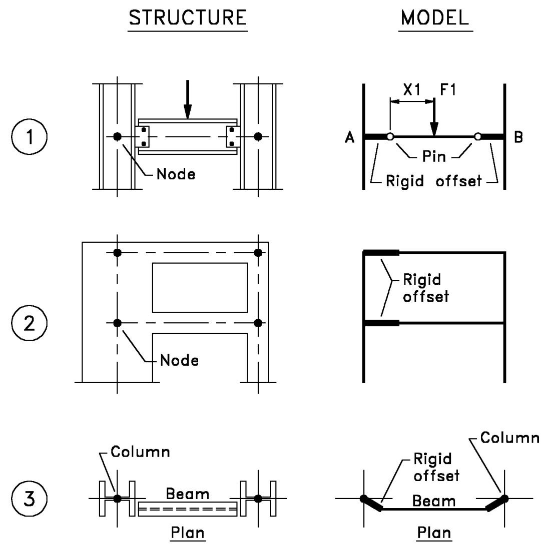
Node mass input is only relevant in dynamic analysis. Accelerations specified in static analysis will not result in inertial loads being applied at nodes where mass has been input.

Use the **Structure > Attributes > Node** command to input node mass at one or more nodes.

Rigid Member Offsets

If designing a structure under steel design code rules for "simple construction" you must use rigid member offsets.

Where members meet at the joints in a structure, there may be little distortion in the part of the structure that is common to more than one member. This is the "joint block" region. If the member cross-section dimensions are large, the very stiff joint block region may be relatively large. When rigid member offsets are used to model the joint block region the end of the member is considered to be connected to the adjacent node by an infinitely rigid link. The diagram below shows typical applications of rigid member offsets.



RIGID MEMBER OFFSETS

Rigid member offsets may be input using the **Structure > Attributes** sub-menu. The extent of each rigid member offset is input by specifying the distance, measured in either the global or local axis directions, from the node to the end of the member. The following data is required:

dXA	X offset from node "A" to end of member.
dYA	Y offset from node "A" to end of member.
dZA	Z offset from node "A" to end of member.
dXB	X offset from node "B" to end of member.
dYB	Y offset from node "B" to end of member.
dZB	Z offset from node "B" to end of member.

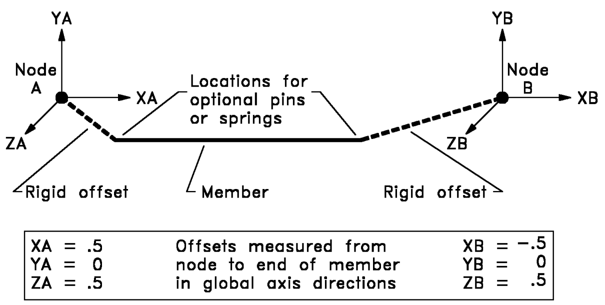
The connection between the node and the rigid member offset is rigid. Any releases specified for the member will be inserted between the end of the member and the rigid offset, not at the node. Similarly, if a semi-

rigid connection is specified adjacent to a rigid member offset, springs will be inserted between the end of the member and the rigid offset.

Loads can be applied to the flexible part of the member only. All load offsets are measured from the "A" end of the member as shown in example (1) in the diagram above. Many rigid member offsets are simply extensions of the member centre-line but they do not have to be. For example, where beams are aligned in plan with the faces of columns, the rigid offset will be cranked as shown in example (3) above.

The diagram below shows how rigid member offsets are specified with respect to the global axis directions.

Note: The orientation of the local axes used for specifying offsets may be changed by the offsets.



RIGID MEMBER OFFSETS SPECIFIED IN GLOBAL AXES

Semi-Rigid Connections

In some structures there may be partial continuity between the ends of a member and the adjacent nodes. You may model this situation with semi-rigid connections, which may be input using the **Structure > Attributes** sub-menu. Any member for which semi-rigid connections are specified is considered to have axial and rotational springs inserted between the end of the member and the adjacent node or rigid offset. The spring stiffnesses are specified with respect to member axes and may be entered independently at each end. The following data is required:

KxA	Spring constant at “A” for displacement along member *x* axis.
KRyA	Spring constant at “A” for rotation about member *y* axis.
KRzA	Spring constant at “A” for rotation about member *z* axis.
KxB	Spring constant at “B” for displacement along member *x* axis.
KRyB	Spring constant at “B” for rotation about member *y* axis.
KRzB	Spring constant at “B” for rotation about member *z* axis.

Units are those specified for spring supports.

An error will be reported if a member contains a pin and a semi-rigid connection at the same end.

Sometimes, friction joints can be modelled approximately by using appropriate spring constants (possibly determined from tests). A pinned connection has a spring constant of zero and a rigid connection has a spring constant of infinity.

The diagram above shows the locations of the spring elements for a member that has rigid offsets.

Note: Ill-conditioning will result if a very large value is used to specify an “infinite” stiffness – input “R” to denote a rigid connection DOF.

Master-Slave Constraints

You may input master-slave constraints using the **Structure > Attributes** sub-menu. This feature lets you impose relationships between displacement components at different nodes. Displacement components at a *slave* node may be constrained to conform to those of other nodes, designated *master* nodes.

One of the main applications of the master-slave feature is in modelling floors considered to be rigid in their own plane. One node on the floor is specified as the master node and all other nodes at that level are slaves for the in-plane DOF. In effect, in-plane DOF are not required for each slave node because the three components of displacement in the horizontal plane can be determined from the in-plane displacements of the master node.

The master node number is specified for each of the selected slave nodes:

MNX	Master node for X translation of slave node.
MNY	Master node for Y translation of slave node.
MNZ	Master node for Z translation of slave node.
MNRX	Master node for RX rotation of slave node.
MNRY	Master node for RY rotation of slave node.

MNRZ	Master node for RZ rotation of slave node.

Set the master node number to zero if the corresponding DOF is not constrained.

The following rules apply to master-slave constraints:

- There may be any number of master nodes.
- Master DOF may not be slaves to any other DOF. (A master node may appear as a slave node provided that the DOF specified as masters are not themselves slaves).
- Slave DOF may not be specified as supports (fixed, or elastic).
- Loads may be applied at slave DOF.

The node displacements and member forces are reported for slave nodes. The number of DOF in the analysis is reduced by the number of constraint relationships. For large structures this may result in some savings in the size of the structure stiffness matrix and in the time required for solution.

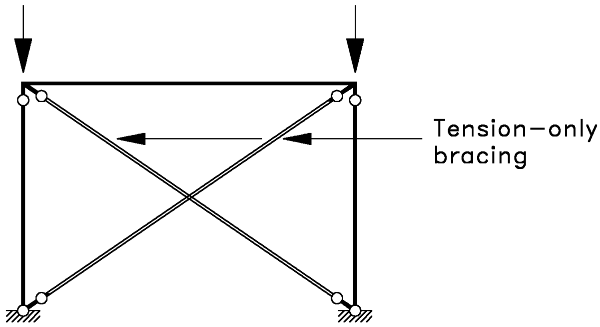
Tension-Only & Compression-Only Members

Members may be marked as *tension-only* or *compression-only* using the **Structure > Attributes > Member Type** command. During non-linear analysis, the axial extension of these members is checked at each iteration and if it is found to be inconsistent with the specified member type, the stiffness of the member is set to zero for the next iteration (i.e. the member is ignored). *Note that any eliminated member may be reinstated in subsequent iterations.*

Tension-only members are often used to model X-bracing members that are so slender that they can support negligible compression. Compression-only members may be used to model supports where “lift-off” occurs under some conditions. As shown in the diagram below, tension-only members may be used to model compression-only elements (and vice versa).

Sometimes, the deletion of a tension-only or compression-only member will cause a structure to become unstable. For example, in the structure shown below, both bracing members would be deleted, causing the structure to be unstable. A simple solution to this particular problem is to apply a horizontal load to produce a small tension in one bracing member or to preload both with a small tension (as would probably occur in practice).

Note: During non-linear analysis the automatic deletion of members in which the axial force would exceed the Euler load can cause a structure to be unstable. See “Troubleshooting Non-Linear Analysis” on page 254.



TENSION-ONLY BRACING

Cable Members

When this advanced option is available the member type may be set to *cable*, using the **Structure > Attributes > Member Type** command. The catenary formulation of the cable element permits the accurate computation of the equilibrium position of each cable under load. On input, the unstrained length, which defaults to the chord length, may be specified for each cable member. Variation of the unstrained length changes the initial tension of the cable. Cable members may be subjected to acceleration loads, uniformly distributed loads in any global axis direction, temperature loads, and axial distortions. Additional nodes may be introduced within cables if it is necessary to apply concentrated loads. The initial tension determined by the unstrained cable length is taken into account automatically.

Cable members have axial tension only – no other member force components exist. Member end releases are not permitted in cable members.

Sag is reported in the plane of the displaced cable in the direction of the resultant load.

Note: When using the **Query > Member Displacements** command the global values tabulated in the output window for a cable are not displacements, but coordinates of the displaced cable. The sag is also shown. Global displacement values for a cable are shown in the analysis report when appropriate options are selected.

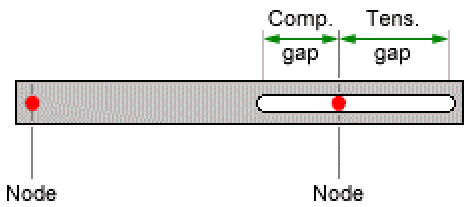
Gap & Fuse Members

When this advanced option is available the member type may be set to *gap*, *brittle fuse*, or *plastic fuse*, using the **Structure > Attributes > Member Type** command. During non-linear analysis, the behaviour of these members is determined as a function of extension or internal axial force. *Gap and fuse members in compression do not buckle. Fuse members have a moment release at each end.*

Gap Member

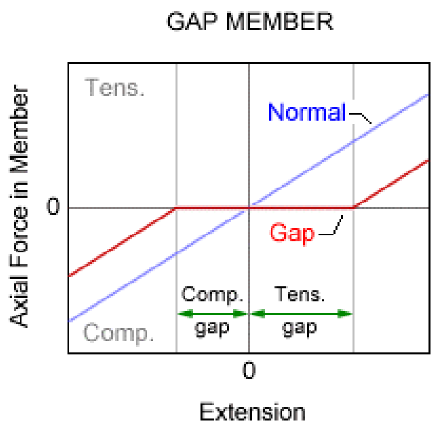
Gap members may be used to model situations where one part of a structure bears on another after a certain relative displacement.

Member properties include two non-negative values, a *compression gap* and a *tension gap*. The compression gap is the amount by which the node-to-node distance may shorten before any compression is induced and the tension gap is the amount by which this distance may lengthen before any tension is induced. A slotted member, whose behaviour can be modelled by the gap member, is shown below.



GAP MEMBER

As the member extension goes beyond either limit the axial force in the gap member increases linearly from zero at the limit. The diagram below shows an axial force-extension graph for a gap member compared to that for a similar normal member.



FORCE/EXTENSION DIAGRAM FOR GAP MEMBER

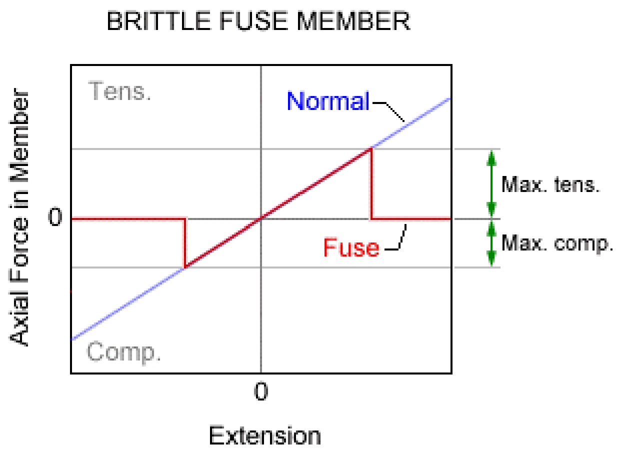
Brittle Fuse Member

Brittle fuse members may be used to model members that fail in a non-ductile manner (e.g. by rupture or buckling) at a specified limiting axial force.

Member properties include two non-negative values, a *maximum compressive force* and a *maximum tensile force*. In a brittle fuse member the axial force becomes zero if axial compression reaches the maximum compressive force or if axial tension reaches the maximum tensile force. In the brittle fuse member the internal axial force becomes zero after it has reached either limiting value. Brittle fuse members cannot buckle.

Note: You may use a fuse member with large values for Pcomp and Ptens to model members that cannot buckle.

The diagram below shows an axial force-extension graph for a brittle fuse member compared to that for a similar normal member.



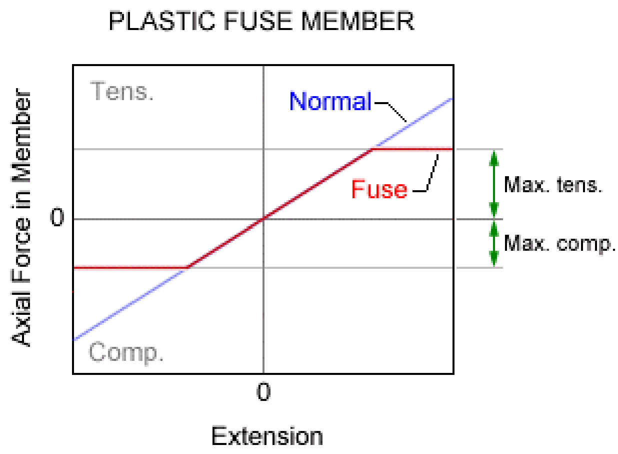
FORCE/EXTENSION DIAGRAM FOR BRITTLE FUSE MEMBER

Plastic Fuse Member

Plastic fuse members may be used to model members that fail in a ductile manner at a specified limiting axial force (i.e. an elasto-plastic member).

Member properties include two non-negative values, a *maximum compressive force* and a *maximum tensile force*. In the plastic fuse member the internal axial force remains constant after it has reached either limiting value. Plastic fuse members cannot buckle.

The diagram below shows an axial force-extension graph for a plastic fuse member compared to that for a similar normal member.



FORCE/EXTENSION DIAGRAM FOR PLASTIC FUSE MEMBER

Note: Analysis of structures containing gap or fuse members may fail if the structure becomes unstable when a member of this type loses its axial stiffness. You may be able to overcome this kind of problem by use of the *displacement control* analysis parameter or by adding very flexible springs or dummy members to keep the structure stable while gap members are not resisting axial force.

Loads on Non-Linear Members

Non-linear members may be removed automatically from the structural model during analysis. Any loads applied to such members are automatically transferred to the end nodes, preserving equilibrium of the applied loads.

Load Case Titles (CASE)

The load data for a job consists of one or more primary load cases and optional combination load cases. Each load case is identified by a unique load case number. The load case number is used for the selection of load cases throughout the Microstran system – in the plotting, reporting, and design modules and for the definition of load combinations. If a load case is deleted, you must ensure that no remaining combination load case refers to it. Each load case is described with a line of up to up to 50 characters of text. This title is used to identify the load case in reports and graphical output.

New Load Case

Click the New Load Case button or select the **Loads > Add Case / Edit Title** command to input the name of a new load case or change the name of an existing load case.

Acceleration Loads (GRAV)

Acceleration or gravitational (GRAV) loads are caused by the effects of acceleration or a gravitational field acting on the mass of the structure. Microstran computes acceleration forces in each global axis direction for each member as the product of the mass of the member and the respective component of acceleration.

GX	Global X component of acceleration.
GY	Global Y component of acceleration.
GZ	Global Z component of acceleration.

Accelerations must be in units consistent with those used elsewhere in the input.

Structure self-weight may be taken into account by specifying an acceleration of –g in the vertical axis direction. For example, if m and kN units are being used, you would specify a value of -9.81 m/sec^2 for the acceleration component in the Y direction (assuming that the global Y axis is the vertical axis).

After selecting an input load case, you may input an acceleration load using the **Loads > GRAV Load** command to specify the acceleration components. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command.

Note: The GRAV load does not create inertial loads for the additional mass usually input for dynamic analysis.

Node Loads (NDLD)

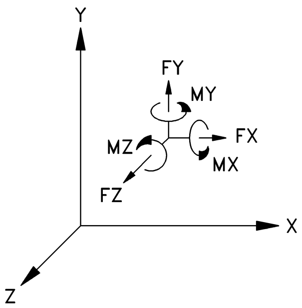
Node loads are loads applied directly to the nodes of the structure. The load on each loaded node is defined by the components of force and moment in the global axis directions.

FX	Force acting in X direction.
FY	Force acting in Y direction.
FZ	Force acting in Z direction.
MX	Moment acting about X axis direction.
MY	Moment acting about Y axis direction.

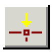
MZ	Moment acting about Z axis direction.

The applied force components are positive when acting in the direction of the global axis. The applied moment components are positive when acting clockwise about the global axis (as viewed along the axis).

Positive node forces and moments are shown in the diagram below.



NODE LOADS



Input Node Loads

After selecting an input load case, you may input node loads by clicking the Input Node Loads button or selecting the **Loads > Node Loads** command to specify the load components. You may also right-click on a node and choose **Edit Node Loads** from the context menu. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command.

Member Loads (MBLD)

Concentrated and distributed forces and couples may be applied at any point in a member. Each member load is specified by the form of the load and parameters describing the intensity and location. Loads may be defined with respect to either the global or the member axes. Where the member has rigid offsets load distances are measured from the end of the member, not the node.

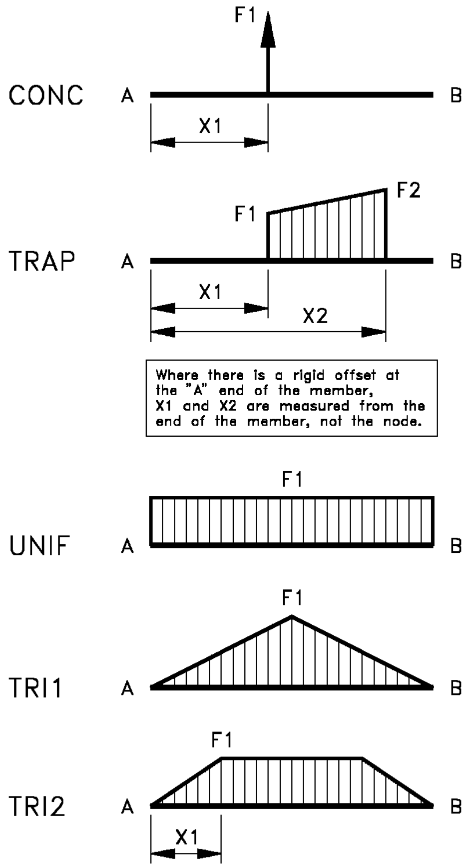
Load Form

A four character mnemonic to define the form of the load:

CONC	Concentrated (point) load.
TRAP	Distributed load over part or all of the member whose intensity may vary linearly.
UNIF	Distributed load of uniform intensity over the full length of the member.
TRI1	Triangular distributed load with zero intensity at the ends of the member ends and maximum at the midpoint.

TRI2	Similar to TRI1 but with the apex symmetrically truncated.

The different forms of member loads are shown in the diagram below.



MEMBER LOAD FORMS

Load Type

A two-character mnemonic to define the load as a force or couple and specify the direction of its action.

FX	Force acting in X direction.
FY	Force acting in Y direction.
FZ	Force acting in Z direction.
MX	Moment acting about X axis direction.
MY	Moment acting about Y axis direction.
MZ	Moment acting about Z axis direction.

The applied force components are positive when acting in the direction of the associated axes. The applied moment components are positive when acting clockwise about the associated axis (as viewed along the axis).

Load Axes

A two-character mnemonic to define the axis system in which the load is described.

GL	The direction of the load is referred to a global axis.
LO	Direction of the load is referred to a member (local) axis.

Load Scale

A two-character mnemonic to define the method of specifying offsets from the “A” end of the member to the load reference points.

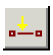
LE	Offsets are in length units.
FR	Offsets are fractions of the member length.

Load Parameters

F1	Load intensity (load per unit length) at the first reference point, or the magnitude of a concentrated load.
X1	Offset (measured along the member) from end “A” of the member to the first reference point.
F2	Load intensity (load per unit length) at the second reference point.
X2	Offset (measured along the member) from end “A” to the second reference point.

Not all of the above parameters are required to describe some load forms. The load forms and the parameters required to define them are shown in the diagram above. The loadings shown are general and may represent forces or moments acting in the direction of any of the global or member axes.

The Member Load dialog box allows you to specify offsets measured from the “B” end of the member. However, offsets are stored internally and represented in the archive file as offsets from the “A” end of the member.

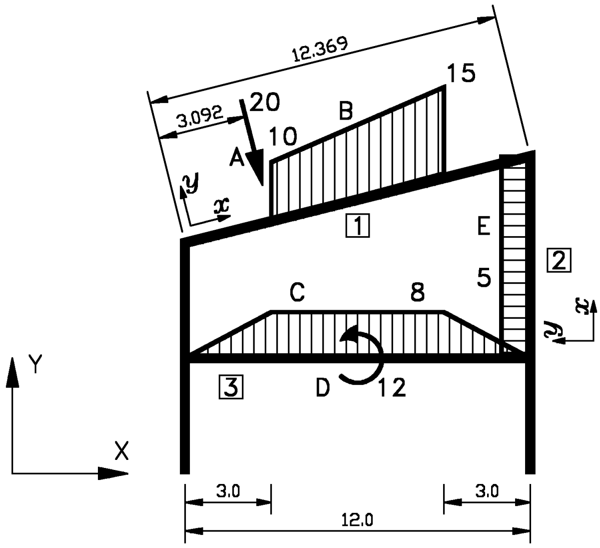


Input Member Loads

After selecting an input load case, you may input member loads by clicking the Input Member Loads button or selecting the **Loads > Member Loads** command.You may also right-click on a member and choose **Input Member Loads** from the context menu. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command.

Member Load Example

Some member load examples are shown in the diagram below. The member load entries required to define these loads are also shown. Each load is represented by a line of data starting with “MBLD”. This is the format used to describe loads in the archive file.



MEMBER LOAD EXAMPLES

Load A:
```
MBLD  1 CONC FY LO FR   -20    .25
```
or
```
MBLD  1 CONC FY LO LE   -20  3.092
```
Load B:
```
MBLD  1 TRAP FY GL FR   -10    .25   -15    .75
```
or
```
MBLD  1 TRAP FY GL LE   -10  3.092   -15  9.277
```
Load C:
```
MBLD  3 TRI2 FY LO FR    -8    .25
```
or
```
MBLD  3 TRI2 FY GL FR    -8    .25
```
Load D:
```
MBLD  3 CONC MZ LO FR    12     .5
```
Load E:
```
MBLD  2 UNIF FY LO       -5
```
or
```
MBLD  2 UNIF FX GL        5
```
Member loads may be defined in a variety of ways. Some loads may be described in either the global or member axis systems and it is advisable to use whichever is more convenient. *Where possible, loads should be*

input with respect to the global axes because the load direction is then evident without reference to member axis orientation. Fractional scaling is particularly useful where loads occur at easily defined intermediate points such as the midpoint or the third-points of the member.

Member Distortions (DIST)

Member distortions, in the form of linear or angular discontinuities, may be introduced at any point in a member. These may be used to model "lack of fit" or prestress. Each distortion is specified as a member load of form DIST. The parameters required to specify the distortion are listed below.

By the Müller-Breslau Principle, influence lines can be made for a structure by plotting the distorted shape of the structure with a unit distortion applied at the point where the influence is required.

Distortion Type

A two-character mnemonic to define the distortion as a linear or angular discontinuity and specify its direction:

FX	Axial discontinuity.
FY	Transverse discontinuity parallel to y axis.
FZ	Transverse discontinuity parallel to z axis.
MX	Angular discontinuity – rotation about x axis.
MY	Angular discontinuity – rotation about y axis.
MZ	Angular discontinuity – rotation about z axis.

Distortion Scale

A two-character mnemonic to define the method of specifying the location of the discontinuity.

LE	Offsets are in length units.
FR	Offsets are fractions of the member length.

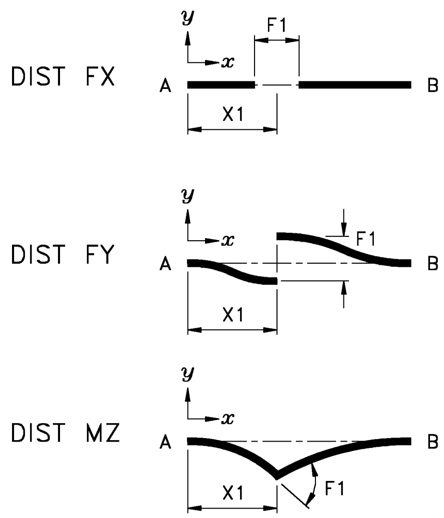
Distortion Parameters

F1	Magnitude of discontinuity in length units for linear distortions and radians for angular distortions.
X1	Offset from end "A" of the member to the location of the discontinuity.

The Member Load dialog box allows you to specify offsets measured from the "B" end of the member. However, offsets are stored internally and represented in the archive file as offsets from the "A" end of the member.

After selecting an input load case, you may input member distortions using the **Loads > Member Loads** command. You may also right-click on a member and choose **Input Member Loads** from the context menu. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command.

Some commonly used distortions and the parameters required to define them are shown in the diagram below. The illustrated distortions are all in the positive sense.

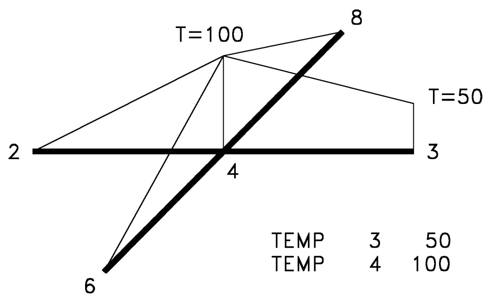


MEMBER DISTORTIONS

Node Temperatures (TEMP)

A temperature differential may be specified at any node in the structure. The temperature differential is the increment of temperature above or below the temperature at which no thermal stresses occur. Temperatures must be in units consistent with those used for the coefficient of thermal expansion.

Every member connected to a node where a temperature differential has been input will be subjected to a constant temperature gradient. The temperature differential varies linearly along the member between the values at the end nodes. In a statically determinate structure the member is free to expand and displacement will occur without the development of any thermal stresses. If the structure is statically indeterminate thermal expansion will be restrained and thermal stresses will result.



NODE TEMPERATURES

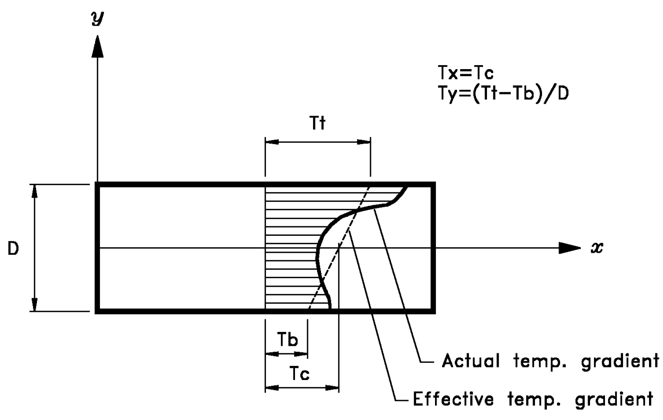
After selecting an input load case, you may input node temperatures using the **Loads > Node Temperatures** command. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command.

Member Temperatures (MTMP)

Member temperature gradients may be input to describe the variation of temperature across the member cross-section. It is assumed that the temperature profile of the cross-section is constant along the length of the member.

Tx	Average temperature differential of the member. If the temperature gradient is linear over the cross-section, Tx will be the temperature differential at the centroid of the section.
Ty	The effective temperature gradient across the member in the y direction. If the member is unrestrained, it causes curvature about the z axis.
Tz	The effective temperature gradient across the member in the z direction. If the member is unrestrained, it causes curvature about the y axis.

The function T(*y,z*) describes the temperature variation over the cross-section of the member. It is assumed that this is constant for the full length of the member. For rectangular sections and linear temperature gradients, the effective temperature gradient is the difference between the temperatures of the extreme fibres divided by the depth of the section.



MEMBER TEMPERATURE GRADIENT

After selecting an input load case, you may input member temperatures using the **Loads > Member Temperatures** command. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command.

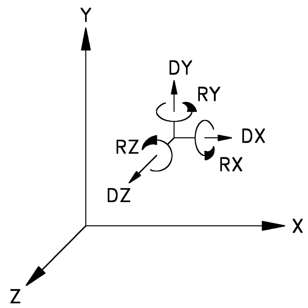
Prescribed Node Displacements (NDIS)

Prescribed displacements are often used to model settlement of supports. Node displacements may be specified in one or more directions at any node. Such prescribed node displacements are defined by giving the components of translation and rotation in the global axis directions. Any DOF where a displacement is prescribed must be defined as a support, i.e. a “1” must be entered into the node restraint code for that DOF.

DX	Translational displacement in X direction.
DY	Translational displacement in Y direction.
DZ	Translational displacement in Z direction.
RX	Rotational displacement about X axis direction.
RY	Rotational displacement about Y axis direction.
RZ	Rotational displacement about Z axis direction.

The translation components are positive when in the direction of the global X, Y, or Z axes respectively. The rotational displacement components are measured in radians and are positive when clockwise about the global X, Y, or Z axes respectively (as viewed along the axis).

Positive translational and rotational displacements are shown in the diagram below.



PRESCRIBED DISPLACEMENTS

After selecting an input load case, you may input prescribed node displacements using the **Loads > Node Displacements** command. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command.

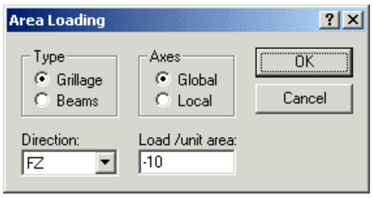
Area Loading on Members

Microstran permits the application of a load uniformly distributed over a selected area. The load is specified as a force per unit area and it is applied as a number of statically equivalent distributed loads on the selected members.

Area Loading

After selecting an input load case, you may input area loading by clicking on the Area Loading button or selecting the **Loads > Area Loading** command. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command.

The Area Loading dialog box allows you to choose Grillage (two-way loading) or Beams (one-way loading).



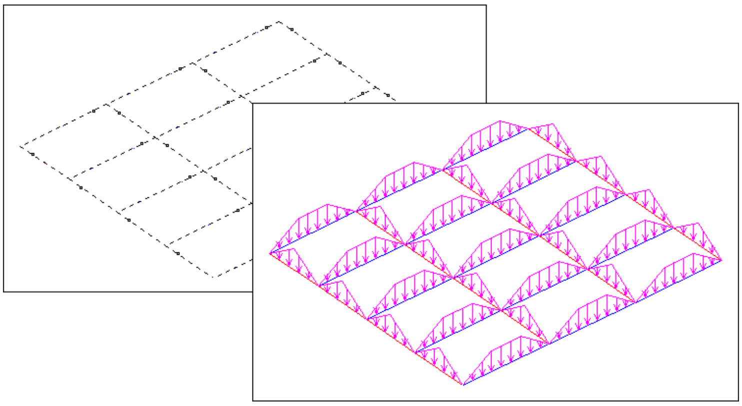
AREA LOADING DIALOG BOX

Two-Way Loading

The tributary area is determined by the outer loop of the selected members. Area loading may be applied simultaneously to several floors or frames, whether overlapping or not. Loads are not applied to members parallel the load direction.

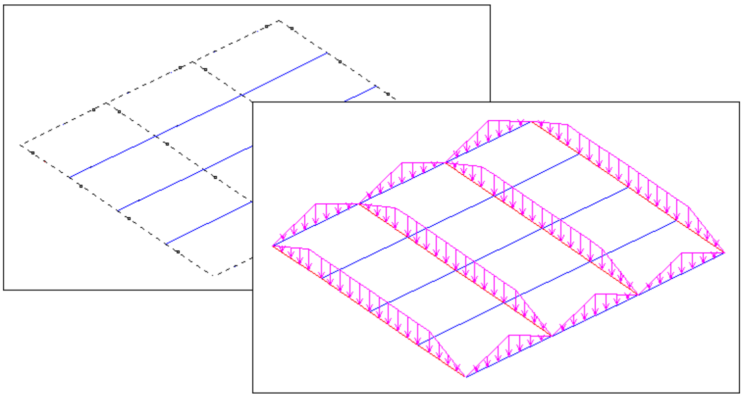
Note: Area loads cannot be applied to any set of members where members cross without intersecting.

The diagram below shows all members of a grillage selected for the application of an area load and the resulting loads on the members forming the grillage.



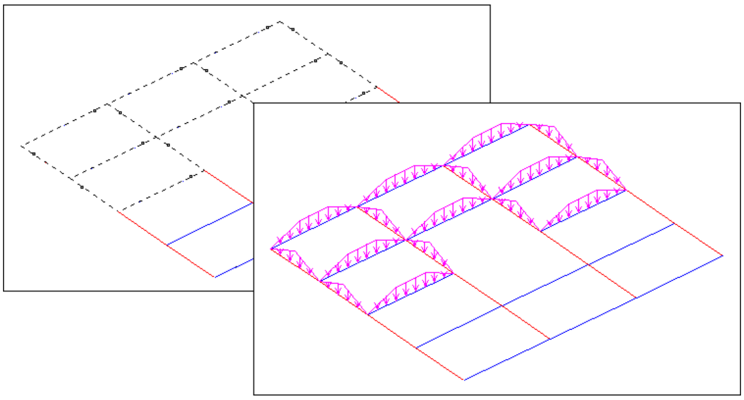
ALL MEMBERS SELECTED

In this diagram some members were not selected for the application of the area loading. Nevertheless, because the area enclosed by the outer loop of selected members is the same, the total area load is the same.



THE SAME TRIBUTARY AREA

The diagram below illustrates how an area load may be applied to a grillage so that a void is not included in the tributary area. The area load must be applied in two steps with the outer loop of selected members in each case not enclosing the opening.

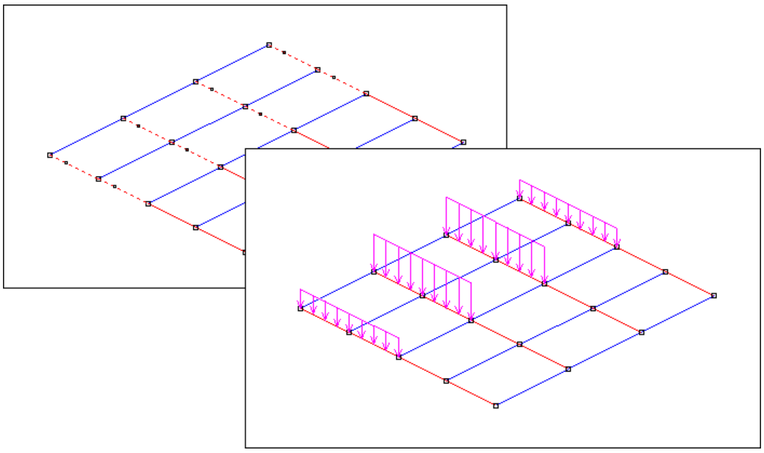


VOID EXCLUDED FROM TRIBUTARY AREA

One-Way Loading

You may select a series of more or less parallel members in a rectangular area to which the load will be applied. Members perpendicular to these should not be selected. The load is applied as a UDL on each member. The magnitude of the UDL is determined automatically from the spacing of the loaded members.

For an irregular layout you should ensure that beams are selected one rectangular "strip" at a time.

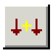


ONE-WAY AREA LOADING

Combination Load Cases (COMB)

Each combination load case requires a load case number, a title, and for each component load case, the load case number and a factor. Component load cases may be either primary load cases or other combination load cases.

Case No.	Load case number. The number of a primary or combination load case. The case number of each component must be less than the current case number.
Factor	Multiplier for load case. The factor by which the specified load case is multiplied before being added to the current combination load case.



Input Combination Case

After selecting an input combination load case, you may input the combination components by clicking on the Input Combination Case button or selecting the **Loads > Case Combinations** command. The input load case is selected by choosing it in the Current Input Case drop-down list box or you may use the **Loads > Select Input Case** command. You cannot input combination components unless the selected load case is a combination load case (i.e. type “C”).

Load Case Templates

A load case template can save time setting up the load cases in a model. You may store a standard set of loads in a load case template and import it into any model to define or add to the load cases.

Use the **File > Configure > Edit Load Case Template** command to edit an existing template or make a new one. Load case template files reside in the Microstran \Lib folder and have an extension of .ltp. After installation from the CD there should be at least one sample load case template file available. A typical load case template file is shown below:

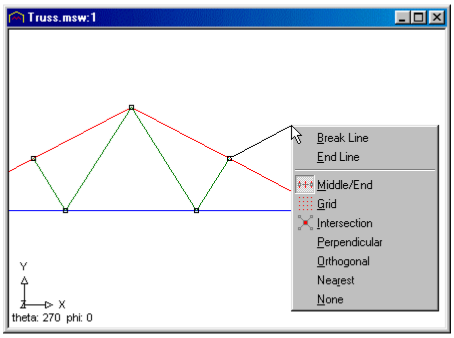
```
CASE  1   DL
GRAV  0  -9.81 0
MBLD 101  UNIF FY GL -10
MBLD 102  UNIF FY GL -10
MBLD 103  UNIF FY GL -10
MBLD 104  UNIF FY GL -10

CASE 2 LL
MBLD 101  UNIF FY GL -20
MBLD 102  UNIF FY GL -20
MBLD 103  UNIF FY GL -20
MBLD 104  UNIF FY GL -20

CASE 11 1.2*DL + 1.5*LL
COMB 1  1.2
COMB 2  1.5

CASE 12 .9*DL
COMB 1  .9

END
```

The load case template is activated for any existing job with the **Loads > Import Load Case Template** command. Any existing load cases that do not conflict with the template remain unchanged. An existing load case having a case number defined in the template is totally replaced by the template load case.

6:Graphics Input

General

Graphics Input is the most efficient input method in many situations. It involves “drawing” a structure on the screen using the mouse or keyboard, and it includes many simple graphical operations, such as copying, moving, rotating, sub-dividing, and erasing. More powerful graphical operations include intersection, extrusion, sub-dividing a member into a curve, and transforming coordinates. In effect, Microstran’s graphical input capability is a CAD system customized for the task of entering structure and load data.

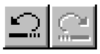


GRAPHICS INPUT

Many Microstran users have found that the few hours required to become proficient at graphical input have been be well rewarded by vastly increased productivity in most aspects of structural analysis and design. A good way to learn Graphics Input is to work through some of the examples in Chapter 12 – “Tutorials”.

Important Note: It is essential that you understand the material in Chapter 2 – “Getting Started” before you can use Graphics Input effectively.

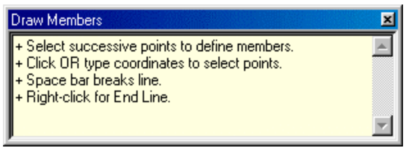
Undo / Redo



Undo restores the job to its status immediately before the last command. It is particularly useful if incorrect data is input in one of the more complex drawing commands. **Redo** reverses the effect of the Undo command when selected immediately afterwards.

The Command Assistant

The Command Assistant is a special help window that appears automatically to provide additional information for many drawing commands. A typical example is shown below. When you no longer require the Command Assistant you may turn it off from the Help menu.



THE COMMAND ASSISTANT

Basic Drawing

Click on the toolbar button to show the grid and then, to start drawing a structure, click on the button. This is the same as selecting the **Structure > Draw Members** command from the menu bar. Notice the *tooltip* "Draw Members" that appears when the mouse cursor crosses this button.

As you initiate the Draw command several things happen:

1. The toolbar button displays in the depressed state, indicating that Microstran is in DRAW mode.
2. "DRAW" is displayed in the status bar at the bottom of the Microstran window.
3. The prompt area of the status bar (on the left) displays the instruction *"Click on first point or enter coordinates"*.
4. The cursor becomes a cross.
5. A help window, the Command Assistant, appears (see above).

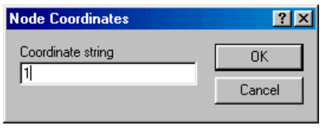
You may now click anywhere in the main window or enter coordinates from the keyboard to locate the "A" node of the first member. Notice that once the first point is specified the prompt changes to *"Click on end point or enter coordinates; press SPACE BAR to break line"*. Select another point and you will have drawn the first member. This point is the "B" node of the first member and the "A" node of the next member. You may continue selecting points to define new members. Refer to "The

Drawing Snap Mode" on page 120 for information on changing the *snap mode*.

Keyboard Entry of Coordinates

Key concept.

There are many situations where the most convenient way to enter a new node is to *type the coordinates*. As soon as you start to type, a dialog box appears to accept your input.



DIALOG BOX FOR ENTERING COORDINATES

Coordinate Systems

You may input coordinates in rectangular, cylindrical, or spherical coordinate systems, using standard syntax or AutoCAD syntax. The format of the coordinate string is described below for each syntax.

STANDARD SYNTAX

- **Rectangular coordinates**
  "X Y Z", where "X", "Y", and "Z" are respectively, the X, Y, and Z coordinates of the point.
- **Cylindrical coordinates**
  "C radius theta h", where "radius", "theta", and "h" are respectively, the radius, horizontal angle, and height of the point.
- **Spherical coordinates**
  "S radius theta phi", where "radius", "theta", and "phi" are respectively, the radius, horizontal angle, and vertical angle of the point.

Trailing zero coordinates do not have to be entered. For example, the point (3,0,0) may be entered as "3". Coordinates must be separated by a space or a comma. Coordinates relative to the last point are preceded by "R" or "r". No separator is required after the "R" or "r".

AUTOCAD SYNTAX

- **Rectangular coordinates**
  "X Y Z", where "X", "Y", and "Z" are respectively, the X, Y, and Z coordinates of the point.
- **Cylindrical coordinates**
  "radius < theta h", where "radius", "theta", and "h" are respectively, the radius, horizontal angle, and height of the point. The last two values must be separated by a space or a comma.
- **Spherical coordinates**
  "radius < theta < phi", where "radius", "theta", and "phi" are

respectively, the radius, horizontal angle, and vertical angle of the point.

Coordinates relative to the last point are preceded by “@”. No separator is required after the “@”.

Breaking the Line

Press the space bar or right-click and choose **Break Line** on the context menu. Notice that the cursor, the status bar, and the button show that Microstran is still in Draw mode. You may now click a new node that is not connected to the last by a member.

Ending the Line

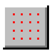
Hit the “End” button or right-click and choose **End Line** on the context menu. Notice the cursor change to the standard arrow. This indicates that the command is finished. The status bar and the button also show that Microstran is no longer in Draw mode.

The Drawing Snap Mode

Key concept.

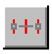
Initially, the status bar displays NONE for the *snap mode*. This means that the coordinates of any node defined by clicking the mouse will be determined by the exact mouse position. The degree of accuracy with which you can position the mouse is limited so the snap mode NONE is rarely used. The first few nodes are usually specified by grid points or entry of coordinates. Thereafter, the Mid/End snap mode is mostly used.

Grid Snap Mode (GRID)



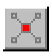
In Grid mode the status bar displays GRID. Grid spacing is initially 1 unit in each global axis direction but you may change it with the **Structure > Drawing Settings > Grid Spacing** command. *When the grid is displayed the cursor snaps to the nearest grid point.* Thus, with the mouse, you can only draw members from one grid point to another. Enter coordinates to specify a point that is not on the grid.

Mid/End Snap Mode (MEND)



When drawing in this mode the cursor snaps to a nearby member end or mid-point. *Most graphical input is done in this snap mode*. When starting a new structure you cannot enter Mid/End snap mode because there are no members to snap to.

Intersection Snap Mode (INTR)



When drawing in this mode the cursor snaps to a nearby intersection of two or more members. A new node is automatically introduced at the intersection point if there is not already a node there. When starting a new structure you cannot enter Intersection snap mode until there are at least two members.

Perpendicular Snap Mode (PERP)

In this mode the cursor snaps to the point on a target member that makes the new member perpendicular to the target member. When starting a new structure you cannot enter Perpendicular snap mode until there is at least one member.

Orthogonal Snap Mode (ORTH)

In this mode you can only draw members in a global axis direction.

Nearest Snap Mode (NEAR)

In this snap mode the cursor snaps to the point on a target member that is nearest to the cursor location.

Changing the Snap Mode "On the Fly"

A very convenient feature is the ability to change the snap mode during a draw operation. For example, you may click the start point of a new member at the end of another while in Mid/End snap mode and then change to Grid snap mode to select the end point. Right-click to display the context menu with its selection of snap modes (see diagram at the beginning of this chapter).

Shortcut Keys

Microstran permits the use of *shortcut keys* to some commands. Shortcut keys are also known as *accelerator keys*. The effect of pressing a shortcut key depends on the context. For example, pressing Delete usually deletes selected members, but in a dialog box it may delete text or do nothing.

Microstran's shortcut keys are listed below:

Main View

Shortcut	Command
←	Viewpoint left *
→	Viewpoint right *
↑	Viewpoint up*
↓	Viewpoint down*
Page Up	Zoom in
Page Down	Zoom out
Ctrl+A	Select all
Ctrl+C	Copy
Ctrl+X	Cut
Ctrl+V	Paste
Ctrl+Z	Undo
Ctrl+Y	Redo
F5	Redraw
Delete	Erase selected members
Home	Zoom to extents/limits
Shift	Enter Dynamic Rotate mode
Esc	Cancel
Enter	Confirm or OK

* May be configured to rotate structure instead of moving viewpoint.

Graphics Input

Shortcut	Command
D	Draw members
E	Erase members
M	Move members
L	Linear copy
P	Polar copy
R	Reflect members
O	Rotate members

S	Sub-divide members
2	Sub-divide members into two
3	Sub-divide members into three

Graphics Input while in Draw Mode

Shortcut	Command
M	Middl/end snap mode
G	Grid snap mode
I	Intersection snap mode
O	Orthogonal snap mode
P	Perpendicular snap mode
Space	Break line
End	End line

OpenGL (Virtual Reality)View

Shortcut	Command
←	Viewpoint Left *
→	Viewpoint Right *
↑	Viewpoint Up*
↓	Viewpoint Down*
Page Up	Zoom In
Page Down	Zoom Out
O	Show global axes
S	Show node symbols
N	Show node numbers
M	Show member numbers
L	Show steel design restraints (LTB)
X	Show connection symbols
C	Show connections
F	Show connection status flags
R	Reverse connection status flags
J	Save .JPG image file
P	Print image
Esc	Revert to initial view

* May be configured to rotate structure instead of moving viewpoint.

The Drawing Plane

The drawing plane is a plane on which nodes are located when you draw in either the Grid or NONE snap modes. For example, when drawing in Grid snap mode with default settings, the drawing plane is X-Y at an offset of zero along the Z axis. This means that all new nodes drawn in Grid or null snap mode have a Z coordinate of zero. Changing the view with any of the **Front View**, **Back View**, **Right View**, **Left View**, or **Top View** commands automatically changes the drawing plane so that it is parallel to the view plane.

Use the **Structure > Drawing Settings > Drawing Plane** command to change the drawing plane as required. If you change the view or the drawing plane so that it (the drawing plane) is at right angles to the view plane (the plane of the screen) you may see the warning message shown below and you may not be able to click a new point.

WARNING THAT DRAWING PLANE IS PERPENDICULAR TO SCREEN

Automatic Removal of Duplicate Nodes and Members

Key concept.

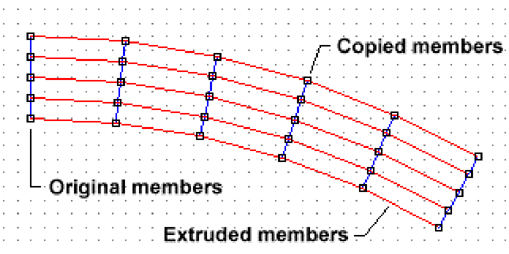
At various stages during graphical input operations, Microstran removes any duplicate nodes or members that are detected. The first node or member to be drawn will remain and any that are superimposed will be removed automatically.

Extrusion

Key concept.

There is a check box for "Extrude nodes" in each of the Linear Copy, Polar Copy, and Reflect dialog boxes. When you perform a copy operation you may "extrude" each copied node into a series of members – in other words, there will be a string of new members lying on the path traced out by each node involved in the copy operation. The member *x* axis is aligned with the direction of extrusion.

Extrusion is a powerful feature that enables you to create many types of structure with minimal effort. For example, you can make a curved bridge deck model by drawing a member representing a cross-beam at one end, sub-dividing it as required to create a node at the location of each girder, and then performing a polar copy with extrusion. There will be a new cross-beam, together with connected girders (longitudinal) for every copy. The diagram below illustrates this procedure.



EXTRUSION OF NODES

Interrupting Commands

Most commands may be *interrupted* by clicking a button on theView toolbar, below. This is helpful in many situations, for example, when drawing a member, and the view required for displaying the "B" node is different from that in which the "A" node is visible. You may interrupt graphical commands to rotate the view, zoom in to a congested area of the model, or pan the view, as required.

VIEW TOOLBAR

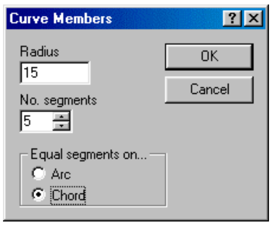
You may also interrupt commands by clicking buttons on the Display toolbar, shown below.

DISPLAY TOOLBAR

The Curve Command

If you complete a curve command that does not have the desired effect you may click the Undo button,  *.*

The **Curve** command is used to sub-divide a member into a number of segments that are chords of an arc. You may select one or more members for this command. When you see the prompt, *Click on point on plane of arc or enter coordinates*, you must either click a point with the mouse or enter the coordinates of a point that, together with the end nodes of the member, defines the plane of the arc. This point does not have to be the centre of the arc but it must lie on the same side of the arc as the centre. For example, if curving a horizontal member into an arch, the point must be below the member in the plane of the arch.

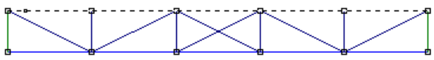


CURVE MEMBERS DIALOG BOX

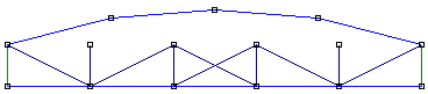
Selecting equal segments on the arc will create new members that are equal in length. *Equal segments on the chord* is the option normally required when curving a chord of a truss – in this case, vertical web members remain vertical after the operation.

The sub-division of the member in the **Curve** command introduces new nodes that are initially on the chord of the arc. At this stage, if any of these nodes are coincident with existing nodes they will be merged with them. The effect of this is that members connected to the pre-existing coincident nodes will be "stretched" so they remain connected to the curved member. This is illustrated in the example below.

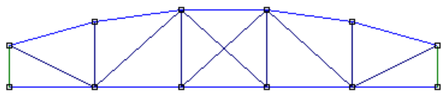
The *single member forming the top chord* is selected for the **Curve** command.



This is the result when 4 segments are selected...



and, when 5 are selected...



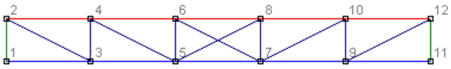
The Stretch Command

If you complete a stretch command that does not have the desired effect you may click the Undo button, .

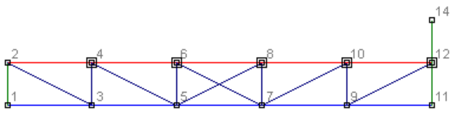
The **Structure > Move > Stretch** command applies a linear transformation to the coordinates of selected nodes. The prompts in the status bar guide you through the necessary steps in this command:

- Select nodes
- Select point as fixed point
- Select point as start point of stretch vector
- Select point as end point of stretch vector

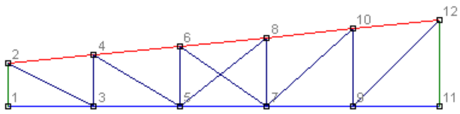
Coordinates of any point may be typed instead of clicking it with the mouse. An example is illustrated below, where the top chord nodes of a truss are “stretched” to introduce a uniform slope from one end to the other.



The *stretch vector* may be defined by selecting two points with the mouse or entering their coordinates. In this example a member is added to represent it. All the nodes to be transformed are highlighted. Node 2 is selected as the *fixed node*.

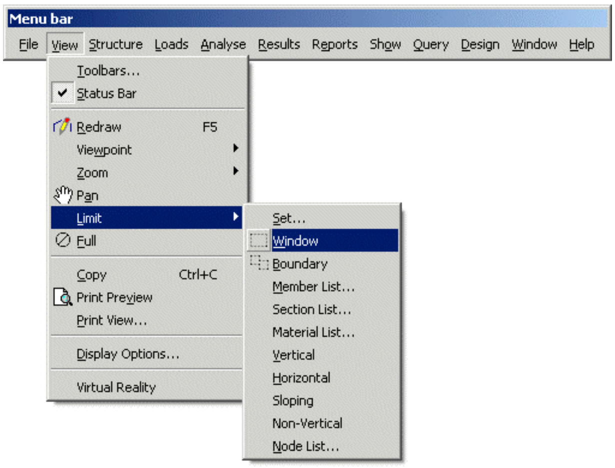


Nodes 12 and 14 are selected to define the *stretch vector*. The diagram below shows the truss on completion of the command.



If you inadvertently click on the wrong node when selecting the fixed node or the start of the stretch vector, you can abort the command by pressing the Esc key.

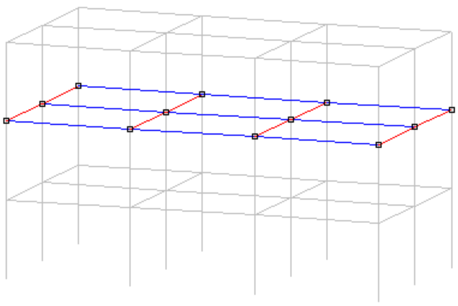
The Limit Command



VIEW > LIMIT > WINDOW

The commands on the **View > Limit** menu allow you to restrict activity to a selected part of the structure. The rest of the structure may be greyed out or hidden from view. This has the advantage that the view you are working on is uncluttered by irrelevant detail and the rest of the structure is *inaccessible* while **Limit** is in effect.

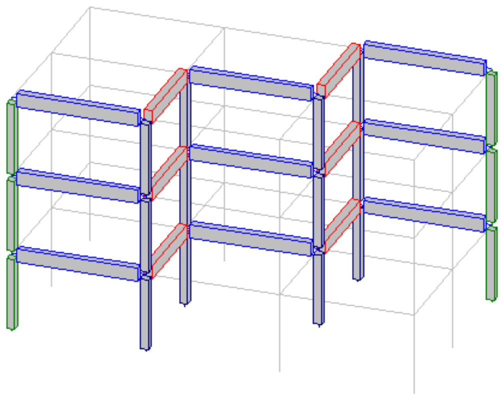
The **Limit > Window** command,  , was used to select one floor of the building structure in the diagram below. To hide the rest of the structure right-click and uncheck **Show Outside Limits**.



LIMIT > WINDOW

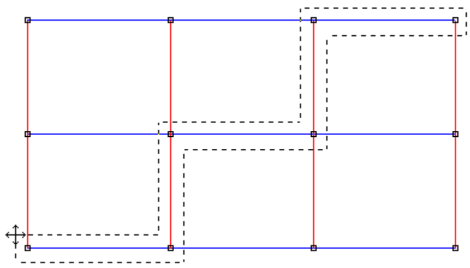
When the **Limit** command is in effect, clicking this button,  , (equivalent to the **View > Zoom > Extents/Limits** command) will zoom the view so that the full structure and the limited part alternately fill the screen.

The **Limit > Boundary** command, , was used to select another part of the same building structure in the diagram below.



LIMIT > BOUNDARY

The selection was made from a plan view by clicking a polygon enclosing the desired part of the structure.

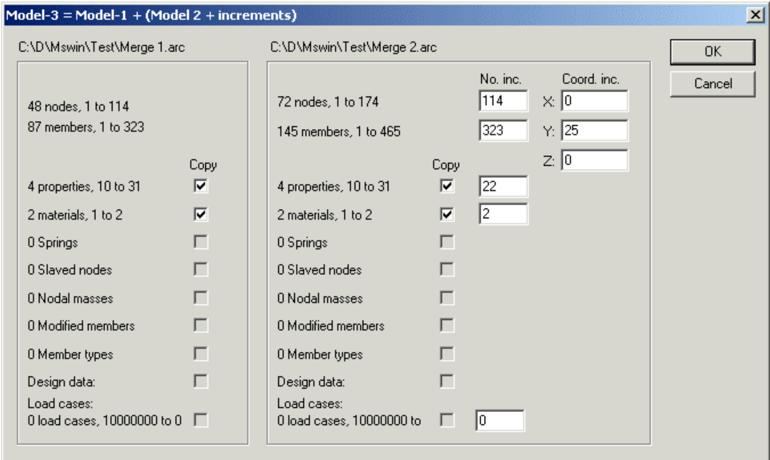


DEFINING THE BOUNDARY

Clicking the Full View button, , reverses the effect of the **Limit** command.

Merging Two Models

The **Structure > Merge Models** command creates a new structural model from two existing models. The current job, if it exists, is closed and the new structure is opened in Microstran. After initiating the command you must select an archive files for each component model and a name for the new model. The dialog box below is then displayed. It shows the statistics of each component model and allows you to specify increments for node, member, and load case numbers so that overlaps in these numbers may be avoided. Any overlap will result in entities with duplicate numbers and the merging of the models will fail.



MERGE MODELS DIALOG BOX

Load Input

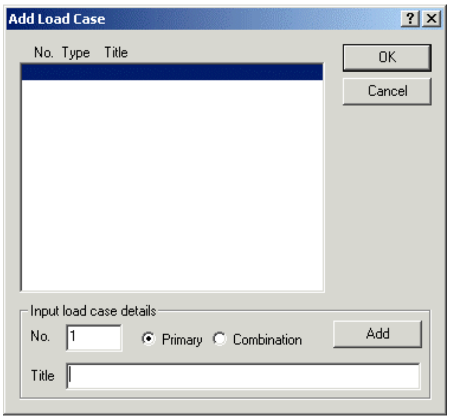
Most load input tasks can be accomplished in Graphics Input with the buttons on the Load Input toolbar (see “Load Input Toolbar Commands” on page 55). Other load input commands are available on the Loads menu (see “Loads Menu Commands” on page 45). Refer to Chapter 5 – “Structure & Load Data” for information on load types not covered below.

Load Case Titles

New Load Case

The first step in entering load data is to define *load cases*, one or more primary load cases and optionally, combination load cases (see “Load Case Titles (CASE)” on page 101).

On clicking the New Load Case button or selecting the **Loads > Add Case / Edit Title** command the dialog box below is displayed. After entering load case details you may click the Add button to transfer the load case to the list box. You can change the title of an existing load case by selecting it in the list box. When you first add a load case you must determine whether it is a primary load case or a combination load case. The type of an existing load case cannot be changed but you can delete a load case and then add it again.

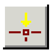


ADDING A LOAD CASE

Selecting the Input Load Case

The Load Input toolbar contains a combo box with the name of the current input load case (see “Load Input Toolbar Commands” on page 55). You cannot input loads when this box is empty. The input load case is selected in the combo box or with the **Loads > Select Input Case** command. Click the button with the red “stop sign” to exit from load input mode.

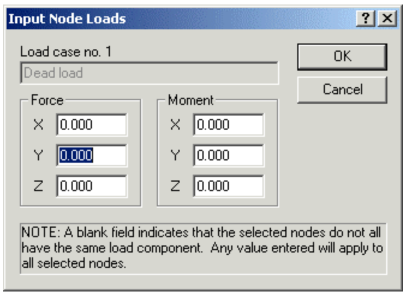
Node Loads



Input Node Loads

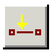
You may input node loads by clicking the Input Node Loads button or selecting the **Loads > Node Loads** command. You may also right-click on a node and choose **Edit Node Loads** from the context menu. If more than one node is selected the specified load components are applied to all selected nodes.

Load components are entered in the Input Node Loads dialog box, shown below. The force component corresponding to the vertical axis is initially selected. A blank box indicates that the selected nodes do not all have the same load component. Any value entered will apply to all selected nodes.



NODE LOAD DIALOG BOX

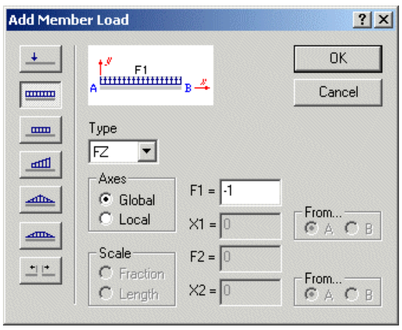
Member Loads



Input Member Loads

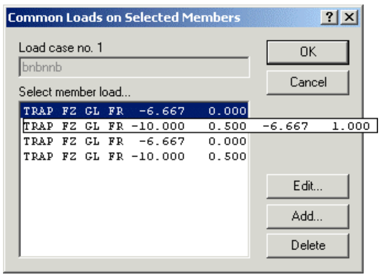
You may input member loads by clicking the Input Member Loads button or selecting the **Loads > Member Loads** command. You may also right-click on a member and choose **Input Member Loads** from the context menu.

Member loads are entered in the Add Member Load dialog box, shown below. The load type is selected and the necessary load parameters are then entered. Clicking the  button provides pop-up help for each item in the dialog box. The load entered will apply to all selected members.



MEMBER LOAD DIALOG BOX

The **Loads > Edit Member Loads** command allows you to change loads that are common to a group of members. This command is also available on the member context menu. All member loads common to the selected members are displayed in the dialog box shown below. You may select one of these and then edit or delete it. You can also add a new member load to all the selected members.



DIALOG BOX FOR EDITING MEMBER LOADS

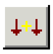
The **Loads > Copy Member Loads** command allows you copy the loads on a single member, for a specified load case, to multiple target members.

Area Loading

AreaLoading

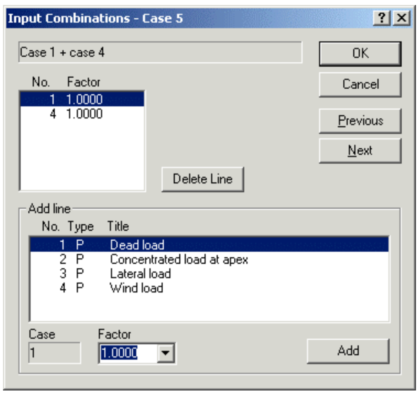
You may input area loading by clicking on the Area Loading button or selecting the **Loads > Area Loading** command – see “Area Loading” on page 112.

Combination Load Cases



Input Combination Case

You may input combination load case components by clicking on the Input Combination Case button or selecting the **Loads > Case Combinations** command. You cannot input combination components unless the selected load case is a combination load case (i.e. type “C”).



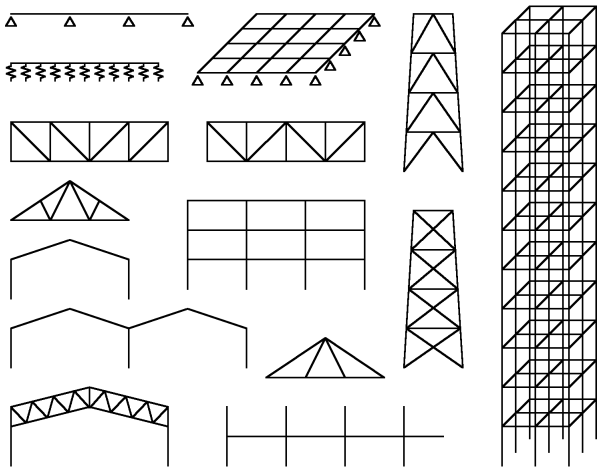
COMBINATION INPUT DIALOG BOX

Combination components are entered in the Input Combinations dialog box, shown above. For each component load case you must enter a factor and then click the Add button.

7:Standard Structures Input

General

Standard Structures Input (SSI) allows you to input structure data for several common types of structure by specifying a small number of defining parameters. All the types of structure shown, and also geodesic domes, can be input with SSI. The extent to which section and material properties may be input with SSI varies from one standard structure to another.



STANDARD STRUCTURES INPUT

Even if SSI cannot generate exactly what is required it is often useful to start the data input with SSI and then add to it or modify it as required with other input methods.

What SSI Does

Job Title, Units and Type

The title for the job may be entered by selecting the **Structure > Title** command**.** SSI determines units, structure type, and the vertical axis *and these should not be changed*. Units are meters, kN, tonnes, and degrees Celsius. Plane frames have the Y axis vertical.

Numbering Scheme

Nodes, members, and sections are numbered to facilitate identification and selection of member groups. In some cases there are gaps in the numbering sequences to permit insertion of additional nodes, members, and sections. The numbering scheme can readily be altered by using the **Structure > Renumber** commands.

Member Releases

SSI makes all members rigidly connected to one another, except for the web members of triangular trusses, which have *z* moment releases at each end. Use another input method to enter any additional member releases that are required.

Sections

For some types of standard structure, SSI allows you to select sections from a section library. In other cases, SSI assigns arbitrary section properties. The desired sections can then be specified with another input method.

Materials

In some cases SSI offers the choice of materials and otherwise assigns the properties of steel or concrete to all members, according to structure type. Material properties can be changed as required with another input method.

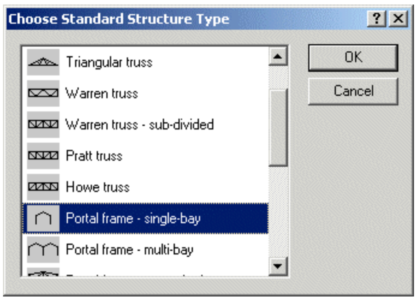
Applied Loads

SSI does not permit the entry of loads. Other input methods must be used to add load data before analysing the structure.

Editing Data Created by SSI

Once you have exited from SSI, the structure data may be changed as required with any other input method, such as Graphics Input or Table Input. *If you go back to SSI to make a change, you will have to enter data again.*

Choosing the Structure Type



CHOOSING THE STRUCTURE TYPE

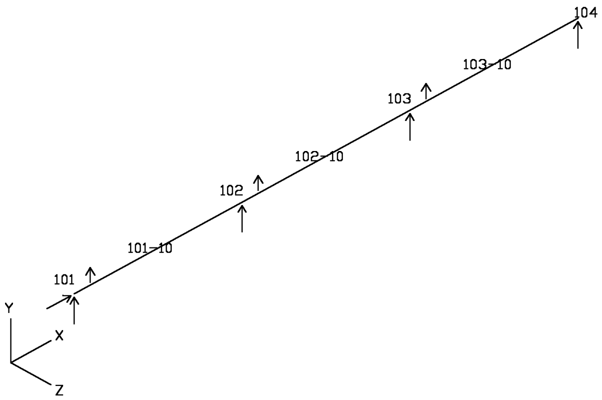
The available structure types are:

- **Continuous beam**
  Single span or multi-span continuous.
- **Beam on elastic foundation**
  Continuous beam of varying section on elastic foundation.
- **Triangular truss**
  Triangular roof truss.
- **Parallel chord truss**
  Parallel chord truss. Several web patterns are available.
- **Single bay portal frame**
  Single bay portal frame, with or without ridge.
- **Multi-bay bay portal frame**
  Multi-bay portal frame. The span and height of each bay may be different. Mono-slope and saw-tooth frames may be input with appropriate choice of input parameters.
- **Trussed rafter portal frame**
  Trussed rafter single bay portal frame. You must choose one of the feasible panel configurations listed for the specified parameters.
- **Sub-frame**
  A structure consisting of a continuous beam with columns above and below. This type of structure is often used for the design of floor slabs in multi-storey buildings.
- **Grillage**
  A rectangular or skew grillage in the global XY plane, Z axis vertical. A range of support conditions is available, including an elastic foundation.
- **2-D frame**
  A rectangular multi-storey plane frame.
- **3-D frame**
  A rectangular multi-storey space frame, Z axis vertical.

- **Trestle**
  An equi-angular braced trestle. Bracing may be in a “K” pattern or an “X” pattern.
- **Geodesic dome**
  Any geodesic dome based on the icosahedron, octahedron, or tetrahedron. You must specify the *frequency* (the number of sub-divisions of the basic polyhedron face) and the *class* (the type of sub-division).

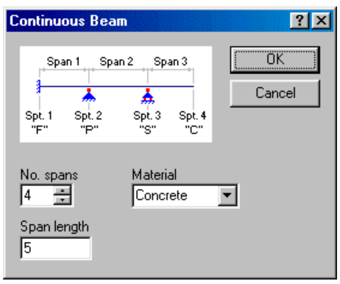
Beams

Continuous Beam



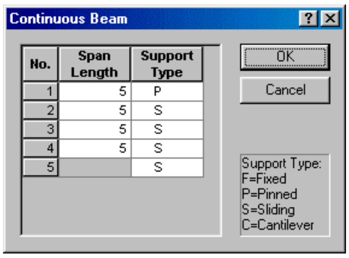
CONTINUOUS BEAM

SSI generates a continuous beam with up to ten spans, each span having the same section properties. At each node the support conditions may be specified as fixed, pinned, sliding, or cantilever (i.e. no support). The structure is a plane frame with the Y axis vertical. The arbitrary section and material properties can be changed with other input methods. The number of spans, the default span length, and the material type are specified in the dialog box shown below.



CONTINUOUS BEAM

The span lengths and support conditions may be varied as required.



CONTINUOUS BEAM – SPAN LENGTHS

Beam on Elastic Foundation

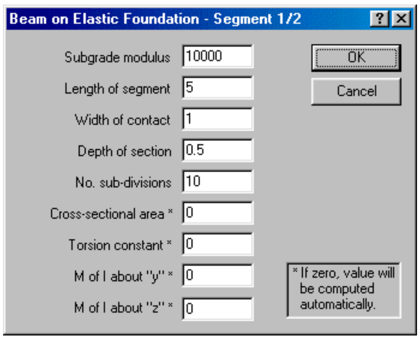
SSI generates a continuous beam with varying section properties on an elastic foundation. The structure is a plane frame with the Y axis vertical.

The beam is input as a number of segments of specified length, width, and depth. Each beam segment is sub-divided into a number of members of equal length (the default value is 10). If section properties are input as zero SSI calculates the values that correspond to the specified width and depth. Material properties for concrete are assigned – these values can be changed with other input methods if necessary.

The subgrade modulus of the foundation material is specified for each beam segment. The foundation material is represented by vertical tension rods fixed at the upper end and connected at the lower end to the nodes in each beam segment. Compression members are not used in order to eliminate the possibility that they may buckle when non-linear analysis is used. SSI calculates section properties for the rods so that the axial stiffness of each rod is equal to the product of the subgrade modulus and the foundation contact area for that rod. *The rods must be specified as tension-only members if it is necessary to take possible lift-off into account. A non-linear analysis is then required.*

Units used for the subgrade modulus must be consistent with units used elsewhere in the job. Where kN and m are used, the units for subgrade modulus are kN/m^3 (i.e. kPa/m).

After specifying the number of segments, you must enter data for each segment. If the geometric properties of the section are entered as zero Microstran will compute them automatically.



BEAM ON ELASTIC FOUNDATION

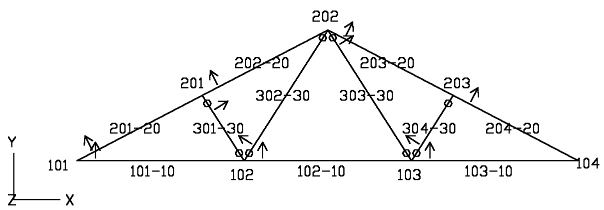
Limitations of the Winkler Foundation
The SSI beam on elastic foundation is a Winkler foundation model, which has limitations in analysing soil/structure interaction. However, if the spring stiffness values can be determined by reference to a known or estimated deflection pattern, reasonable results may be obtained.

Subgrade Modulus

The subgrade modulus varies over a very wide range depending on the soil type and footing geometry. Typical values range from 10000 to 150000 kN/m^3 for granular soils and from 20000 to 100000 kN/m^3 for cohesive soils.

Trusses

Triangular Truss

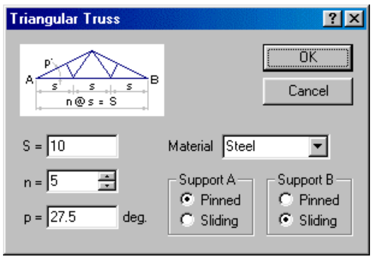


TRIANGULAR TRUSS

SSI generates a symmetrical triangular truss with web members connected to the chords at equal intervals in a “W” pattern. The number of panels in the bottom chord must be an odd number between 3 and 19. One section number is assigned to all top chord members, another to all bottom chord members and another to all web members. The chords are continuous (i.e. there are no member releases in the chord members), while web members are pinned at each end (i.e. they have a *z* moment release at each end). The structure is a plane frame with the Y axis

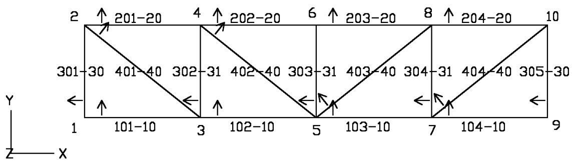
vertical. The arbitrary section and material properties can be changed with other input methods.

The dialog box below is used to enter the parameters defining the triangular truss.



TRIANGULAR TRUSS

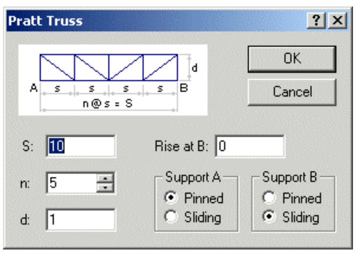
Parallel Chord Trusses



PARALLEL CHORD (PRATT) TRUSS

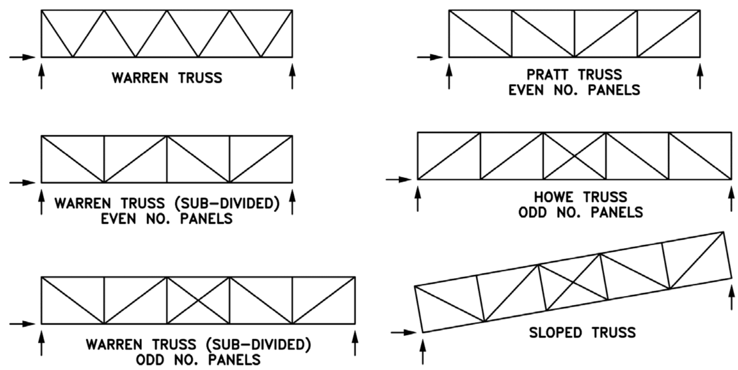
SSI generates the data for four types of parallel chord truss; a Warren truss; , a sub-divided Warren truss, a Pratt, and a Howe truss. If an odd number of panels is specified, all except the Warren truss will have crossed diagonals in the centre panel. The truss will be sloped if a value is entered for the rise between supports. In a sloped truss, the “vertical” web members are at right angles to the chords). The structure is a plane frame with the Y axis vertical. The arbitrary section and material properties can be changed with other input methods.

The dialog box below is used to enter the parameters for a Pratt truss.



PARALLEL CHORD TRUSS

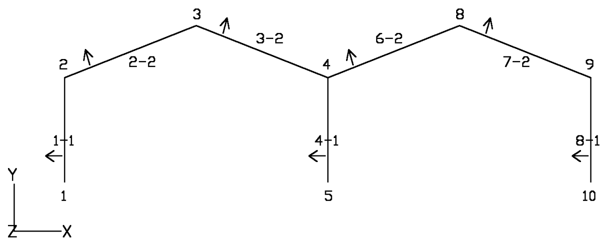
Typical parallel chord trusses are shown in the diagram below.



TYPES OF PARALLEL CHORD TRUSS

Portal Frames

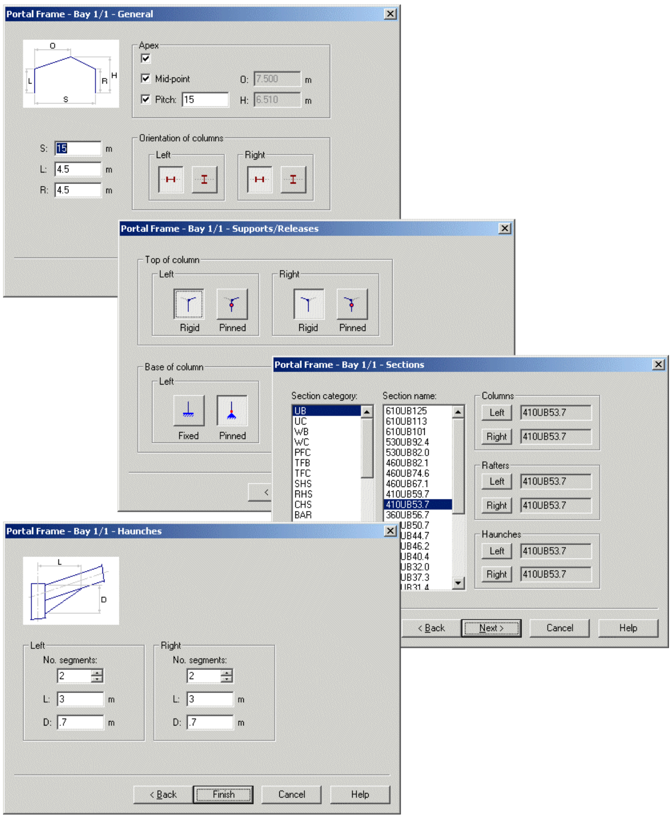
Single Bay & Multi-bay Portal Frames



MULTI-BAY PORTAL FRAME

SSI generates a wide range of single bay and multi-bay portal frames with a minimum of input effort. Each bay of the portal frame may have a ridge located anywhere between the columns, and rafters may have a haunch at column connections. The structure is a plane frame with the Y axis vertical. Sections are selected from the current steel library.

Each bay of the portal frame is described by a *wizard*, each page of which is shown below.



PORTAL FRAME WIZARD

Haunches

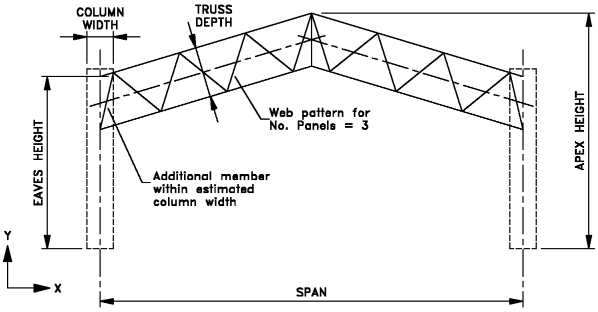
Haunches are fabricated by welding a tee cut from an I section to the rafter at the column connection. A haunch is specified by the name of the section from which the tee is cut and the overall depth of the section at the face of the column (measured vertically). Microstran treats the haunch section as a number of prismatic segments, calculating the section properties at the centre of each segment. In most cases, it is not necessary to specify more than one or two segments to obtain accurate results.

Although SSI does not allow the input of a haunch at an apex, it is simple to achieve by the following procedure:

1. Input the bay with the apex haunch as two bays each without an apex.
2. Specify the desired haunch on the right of the left-hand bay and on the left of the right-hand bay.
3. After exiting from SSI, delete the centre column with Graphics Input.

Trussed Rafter Portal Frame

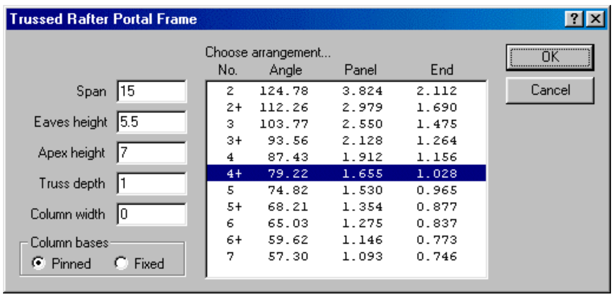
SSI generates data for a single bay symmetrical portal frame with rafters formed by a truss with parallel chords.



TRUSSED RAFTER PORTAL FRAME

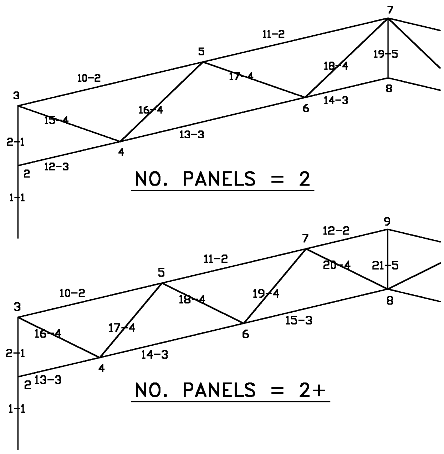
SSI lists a number of feasible panel layouts from which you may choose the most suitable. SSI allows the entry of an estimate for the column width so that the truss geometry can be calculated for the length of the truss being taken as the distance between the column faces. A “dummy panel” is included between the face of the column and the centre-line of the column. If zero is entered for the column width, the length of the truss is taken as the centre-to-centre distance between the columns. The structure is a plane frame with the Y axis vertical.

The chords are continuous (i.e. there are no member releases in the chord members), while web members are pinned at each end (i.e. they have a *z* moment release at each end). The arbitrary section and material properties can be changed with other input methods.



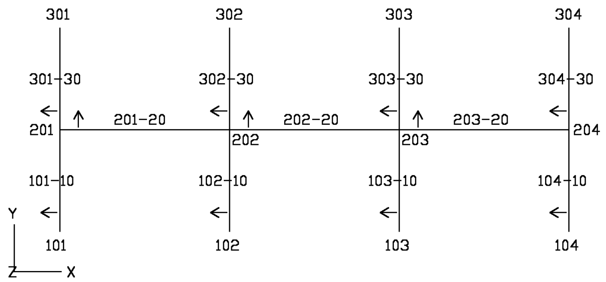
TRUSSED RAFTER PORTAL FRAME
– CHOOSING ARRANGEMENT

The above dialog box is used to enter the parameters defining the structure. All arrangements regarded as feasible are shown in the list box. The arrangement number is the number of panels along the top chord of one side of the truss (not counting the dummy panel that occurs when a non-zero column width is entered). If a “+” is shown next to this number it means that there is an additional part panel on each side of the truss adjacent to the apex. This can be seen in the diagram below.



TRUSSED RAFTER PORTAL FRAME
– NUMBER OF PANELS

Sub-frame

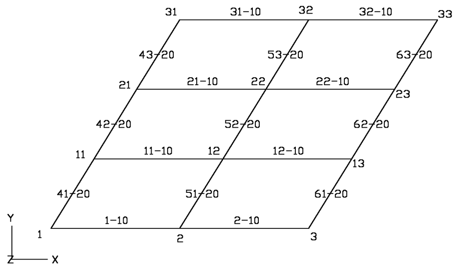


SUB-FRAME

SSI generates data for a sub-frame composed of a continuous beam with up to 10 spans with optional columns above and below the level of the beam. The parameters necessary to specify the sub-frame are the span lengths and the lower and upper column heights, from left to right. Set column height to zero to remove a column – a cantilever will have zero column lengths at the free end. Columns are fully restrained at the levels above and below the beam.

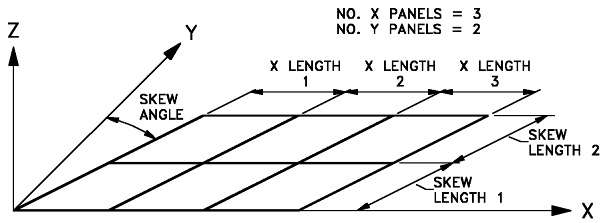
The structure is a plane frame with the Y axis vertical. The arbitrary section and material properties can be changed with other input methods.

Grillage



GRILLAGE

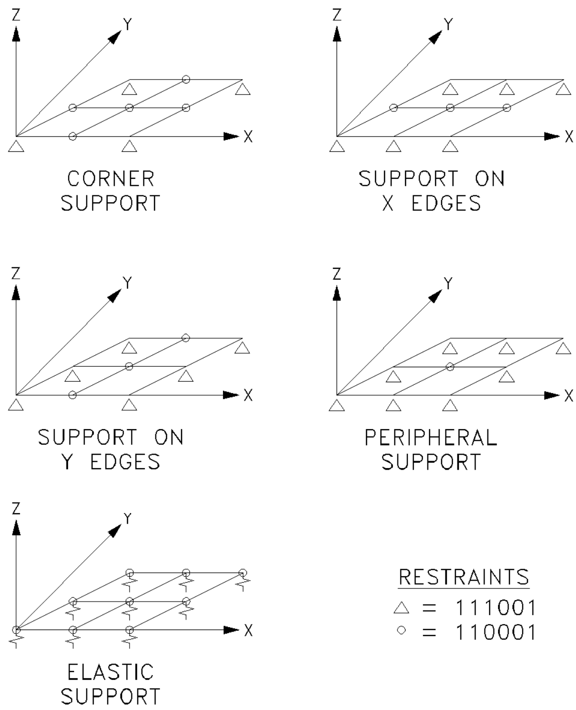
SSI generates data for a grillage in the XY plane with Z vertical. The grillage is specified by the number of panels in each direction, a skew angle, and the length of each panel. One axis of the grillage is in the global X direction and the other, referred to as the skew direction, is rotated clockwise (in plan) from the Y axis direction by the skew angle. The lengths of the skew panels are measured in the skew direction, as shown in the diagram below.



GRILLAGE - DIMENSIONS

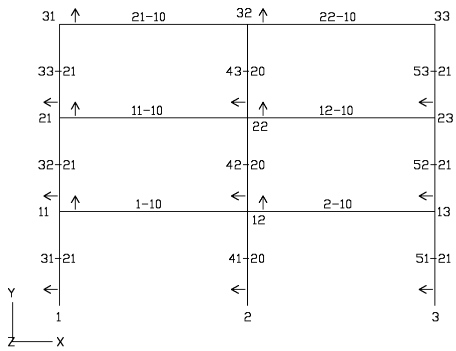
Supports

Several alternative arrangements are available for the supports. When elastic support is selected the subgrade modulus must be entered and the stiffness of each spring support is computed as the product of the subgrade modulus and the contact area for that spring.



GRILLAGE - SUPPORTS

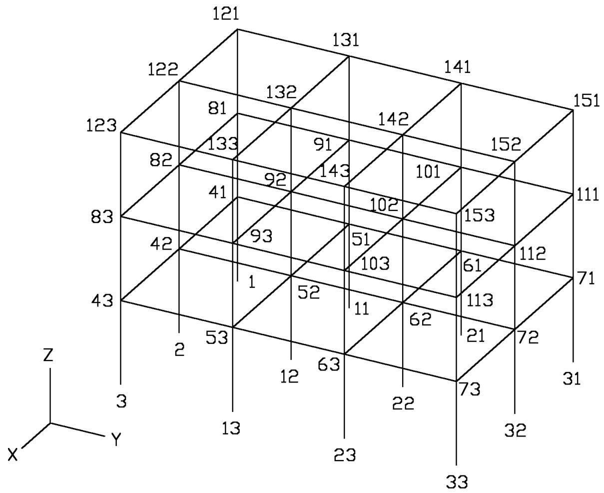
2-D Frame



2-D FRAME

SSI generates data for a rectangular frame in the XY plane. The structure is a plane frame with the Y axis vertical. The input parameters are bay lengths in the X direction, storey heights and support conditions (pinned or fixed). The arbitrary section properties and concrete material properties can be changed with other input methods.

3-D Frame

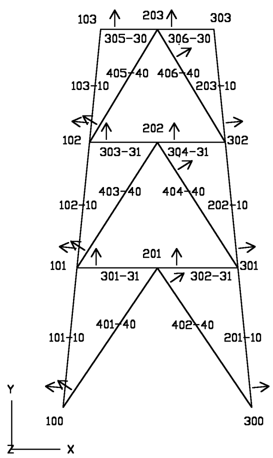


3-D FRAME

SSI generates data for a three dimensional rectangular frame. The structure is a space frame with the Z axis vertical. The input parameters are bay lengths in the X and Y directions, storey heights and support conditions (pinned or fixed). The arbitrary section properties and concrete material properties can be changed with other input methods.

Master-slave constraints are often used to model the rigid in-plane behaviour of concrete floor slabs in this type of structure and may be added if the master-slave option is available (see "Master-Slave Constraints" on page 96).

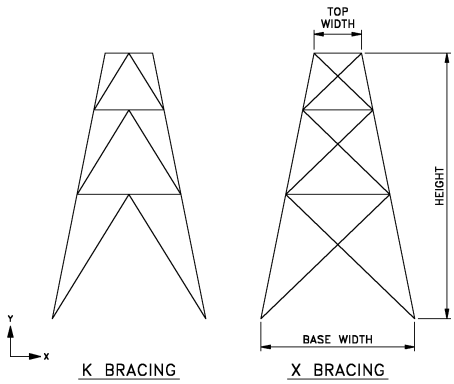
Trestle



TRESTLE

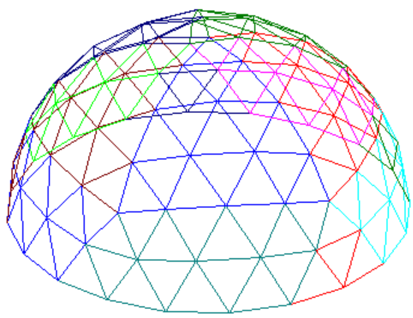
SSI generates data for an equi-angular braced trestle with “K” or “X” pattern bracing. The structure is a plane frame with the Y axis vertical. The base nodes are pin supports. The arbitrary section properties and steel material properties can be changed with other input methods.

In this type of structure, all panels are geometrically similar for structural efficiency and detailing convenience. If a constant panel height is required, the trestle may be input in SSI without taper and then tapered with the **Structure > Move > Stretch** command. The parameters required to define the trestle are shown in the diagram below.



TRESTLE – DIMENSIONS

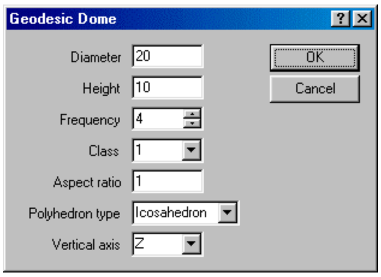
Geodesic Dome



GEODESIC DOME

SSI generates data for geodesic domes based on the icosahedron, the octahedron, or the tetrahedron. The structure is a space frame with either the Y or Z axis vertical. Supports, sections, and materials are not generated – these must be entered with another input method. The members in each polyhedral face have a different section number but they can easily be changed with another input method.

The dialog box below is used to enter the parameters defining the structure.



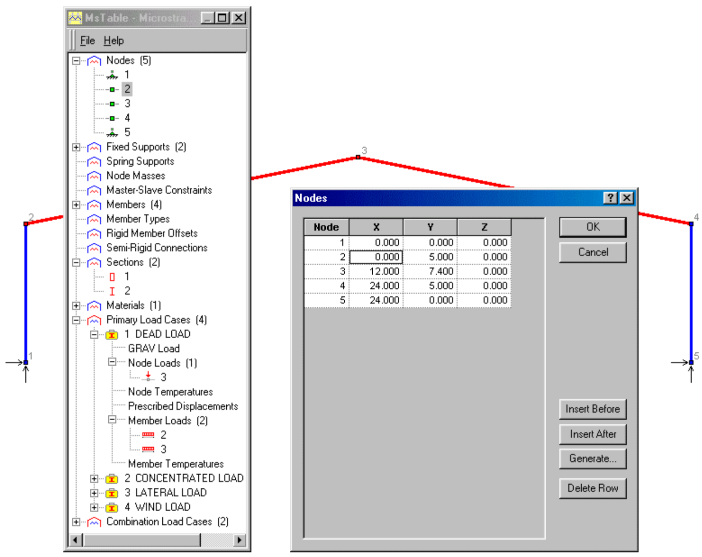
GEODESIC DOME – PARAMETERS

- **Diameter**
  The X and Y diameters of the geodesic surface. The Z diameter is determined by Aspect ratio.
- **Height**
  The height of the dome. If the height is equal to or greater than the vertical diameter, a full sphere (or spheroid) will be generated.
- **Frequency**
  The order of the sub-division of each polyhedral face. Frequency 1 means that the polyhedral faces are not sub-divided.
- **Class**
  The type of sub-division of the polyhedral faces. For Class 1, sub-division triangles are aligned with the edges of the polyhedral faces; for Class 2, triangles are rotated at right angles.
- **Aspect ratio**
  The ratio of the Z diameter to the X and Y diameters. For a spherical surface this value is 1.
- **Polyhedron type**
  A geodesic surface is an approximation to a spherical surface created by the triangulation on the circumsphere of each face of a regular polyhedron. Three regular polyhedrons are available – the icosahedron has 20 faces, the octahedron has 8, and the tetrahedron has 4. All members in the same polyhedral face have the same section number.
- **Vertical axis**
  When Z, the geodesic dome is always circular in plan; when Y, the plan shape will be elliptical if the aspect ratio is not 1.

8:Table Input

General

Table Input allows you to enter, modify, and review all job data in numerical form. It is initiated by clicking a button on the main toolbar or selecting the **Table Input** command on either the Structure menu or the Load menu. This displays a *tree view* of the job data, similar to that in Windows Explorer. An example is shown in the diagram. A heading is shown for each type of entity and if there is a plus sign in a box, you may either click on it or double-click the entity header to expand the list of all such entities. Microstran shows an icon next to each entity to provide a visual cue to the entity type – for example, the icon next to each node shows whether the node is restrained.



TREE VIEW AND TABLE

Double-click on any entity and a table is displayed in a dialog box. The diagram above shows the node table as it appears after double-clicking on node 2 in the tree view. The dialog box has buttons for inserting and deleting rows, and in some cases, for *generating* a series of new entities.

Note that context help (the question-mark button) can provide information on any item in the dialog box.

Double-clicking any item in the tree that has children (as shown by the plus sign in the box) will simply expand or contract the list of children. Double-clicking an item without children will display the corresponding table.

You can use Table Input either for editing an existing job or for making a new job. Data may be entered in any convenient order because comprehensive consistency checks are not carried out until you close Table Input.

Note: Deleting or renumbering an item in Table Input will invalidate other data that refers to it. For example, if you renumber a node, any member that referred to the old node number will now refer to a non-existent node and, unless you adjust the affected data in the member table, the member will be lost when you save your changes. Load and design data may also be invalidated by changes to structure data.

How To Use Tables

Selecting Cells

Tables behave in a similar way to a spreadsheet. In all tables you can move around using the arrow keys on the keyboard or by clicking a cell with the mouse. When a cell is *selected* it is highlighted as shown in the diagram below. Anything you type will completely replace the contents of the cell.

7	3.500	3.500
8	10.500	3.500
9	0.000	0.000

SELECTED CELL

When there is a *caret* in a cell, as shown below, the cell is ready for editing – anything you type will be inserted at the caret.

7	3.500	3.500
8	10.500	3.500
9	0.000	0.000

EDITING CELL CONTENTS

When you click on a cell with the mouse, the resulting state of the cell depends upon the exact location in the cell where you clicked. If you click away from the text in a cell (i.e., to the left of a number), the cell will be selected. If you click *on* the text in the cell the caret will appear where you clicked and the cell will be ready for editing.

You may select multiple cells by dragging the cursor over them (this is useful for deleting a number of rows or copying a range of cells). The diagram below shows part of a table with a rectangular range selected. The top left cell in the selected range has the *focus*. If only one cell is selected, it has the focus.

3	7.000	3.500	0.000
6	7.000	0.000	0.000
7	3.500	3.500	0.000
8	10.500	3.500	0.000
9	0.000	0.000	2.000
10	0.000	3.500	2.000
11	7.000	3.500	2.000
14	7.000	0.000	2.000

SELECTED RANGE

Shortcut Keys

There are several useful shortcut keys. These are:

Ctrl+X	Cut (copy to clipboard and delete)
Ctrl+C	Copy to clipboard
Ctrl+V	Paste from clipboard
Ctrl+Z	Undo the last action

Sorting Rows

Double-clicking on any column header sorts the rows of the table according to the values in the selected column. The first time you double-click, the rows will be sorted into ascending order. Double-click again and the rows will be sorted in the reverse order. Sorting rows in load tables quickly shows the loads with the greatest magnitude.

Insert Before

Click this button to insert a new row *before* the current row (the row containing the cell with the focus). The number of the new entity is usually determined so that gaps in the numbering sequence are preserved, however, if there is not a sufficient gap to the next entity the increment will be 1. If there is no gap in the entity number, no row will be inserted.

Insert After

Click this button to insert a new row *after* the current row. The number of the new entity is usually determined so that gaps in the numbering sequence are preserved, however, if there is not a sufficient gap to the next entity the increment will be 1. If there is no gap in the entity number, no row will be inserted.

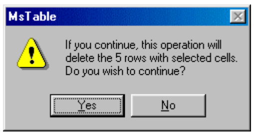
Generate

This button is not available for all entity types. Where it is available, clicking the *Generate* button displays a dialog box so you can specify a generation sequence. On clicking OK in the dialog box, the newly generated entities are inserted into the table.

As you enter data in some of the edit boxes in a generation dialog box, other edit boxes change automatically. For example, in the node generation dialog box, when you change a “To” coordinate the corresponding increment changes. In other edit boxes, automatic update occurs only when the focus leaves the edit box. Click the Update button to force an update of all items and at the same time, validate them. In this way, you can check the results of any change before clicking OK. Invalid data items will be cleared and empty data items will be set to default values.

Delete Row

Where a single cell is selected, clicking the Delete button will remove the row containing the selected cell. If a range of cells is selected, clicking the Delete button will remove all rows with one or more selected cells. Multiple deletions are always preceded by a warning message such as that shown below. No rows will be deleted if you click the No button.



MULTIPLE DELETION WARNING

When you delete a row the focus moves to the next row after the deleted row, so you can quickly delete a number of rows by clicking the Delete button repeatedly. If you delete the last row in a table, the focus moves up to the row before the deleted row. A beep sounds to warn you when this occurs.

Important Note: Design data cannot be input or modified using Table Input. Also, changes to structure or load data may invalidate existing design data. Table Input displays a warning to this effect when you save a job containing design data.

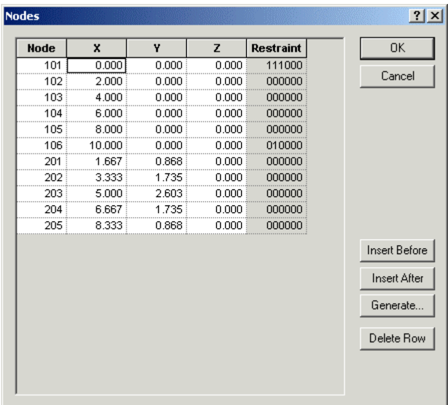
Structure Entities

Below is a list of all the types of structure data that may exist in a Microstran job. You may input or modify all of these entitiy types using Table Input. Those shown bold are documented in detail in this section and the others, which behave similarly, are omitted.

- **Nodes**
- **Fixed supports**
- Spring supports
- Node masses
- Master-slave constraints
- **Members**
- **Member types**
- Rigid member offsets
- Semi-rigid connections
- **Sections**
- Materials

Nodes

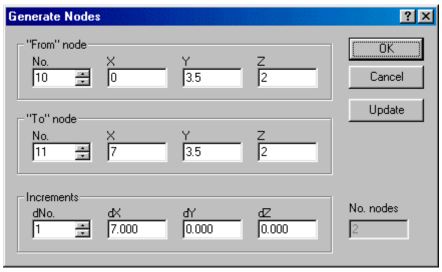
The dialog box below contains the table of node coordinates.



Node	X	Y	Z	Restraint
101	0.000	0.000	0.000	111000
102	2.000	0.000	0.000	000000
103	4.000	0.000	0.000	000000
104	6.000	0.000	0.000	000000
105	8.000	0.000	0.000	000000
106	10.000	0.000	0.000	010000
201	1.667	0.868	0.000	000000
202	3.333	1.735	0.000	000000
203	5.000	2.603	0.000	000000
204	6.667	1.735	0.000	000000
205	8.333	0.868	0.000	000000

NODES TABLE

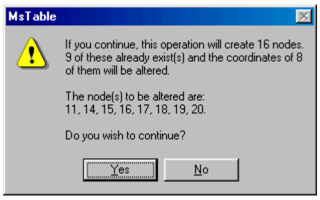
The Generate button displays the dialog box below.



NODE GENERATION

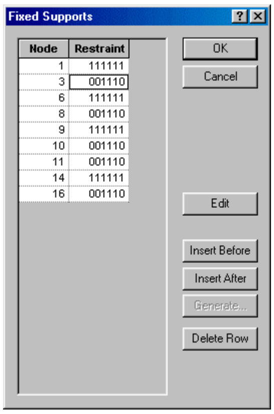
The initial "From" node is the node that was selected in the table. When you select a "From" or "To" node number that exists in the table, the coordinates in this dialog box will be updated automatically to those in the table.

New nodes are generated by interpolating between the "From" and "To" nodes. Any existing node whose number matches a node number in the generated series will be replaced by the generated node. A message is displayed to warn you of any overwriting of coordinates so you can decide whether to proceed or not.



NODE OVERWRITE WARNING

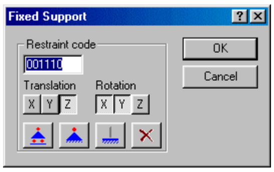
Fixed Supports



Node	Restraint
1	111111
3	001110
6	111111
8	001110
9	111111
10	001110
11	001110
14	111111
16	001110

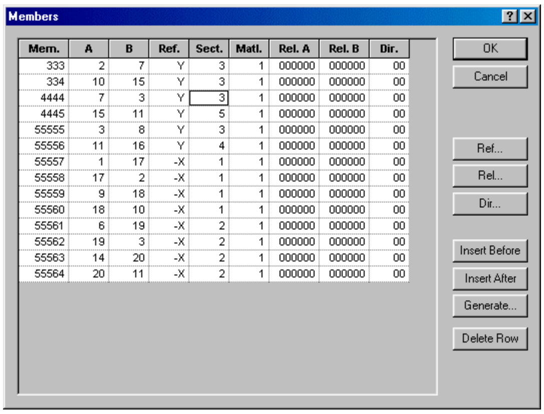
FIXED SUPPORTS TABLE

Clicking the Edit button provides a convenient way to inspect or change the restraint code.



EDITING RESTRAINT CODE

Members



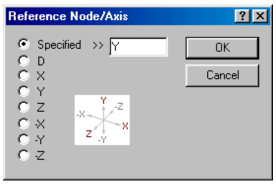
Mem.	A	B	Ref.	Sect.	Matl.	Rel. A	Rel. B	Dir.
333	2	7	Y	3	1	000000	000000	00
334	10	15	Y	3	1	000000	000000	00
4444	7	3	Y	3	1	000000	000000	00
4445	15	11	Y	5	1	000000	000000	00
55555	3	8	Y	3	1	000000	000000	00
55556	11	16	Y	4	1	000000	000000	00
55557	1	17	-X	1	1	000000	000000	00
55558	17	2	-X	1	1	000000	000000	00
55559	9	18	-X	1	1	000000	000000	00
55560	18	10	-X	1	1	000000	000000	00
55561	6	19	-X	2	1	000000	000000	00
55562	19	3	-X	2	1	000000	000000	00
55563	14	20	-X	2	1	000000	000000	00
55564	20	11	-X	2	1	000000	000000	00

MEMBERS TABLE

This dialog box contains the table of members. The Ref., Rel., and Dir. buttons are provided to assist in the entry or display of an item in the respective columns. You may type values directly into any cell in the

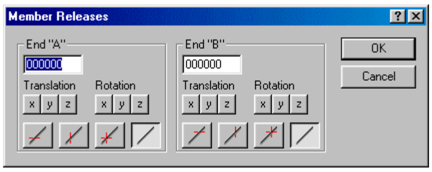
table but clicking one of these buttons displays the corresponding item of the current row in a dialog box.

The Ref. button displays the member reference node/axis dialog box shown below.



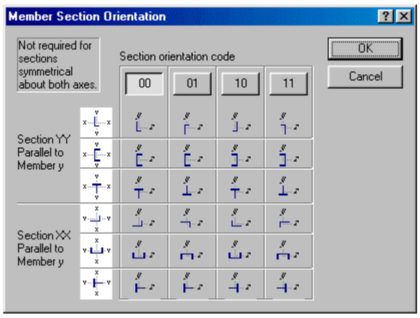
EDITING REFERENCE NODE/AXIS

The Rel. button displays a dialog box, which allows you to inspect or change the selected member releases.



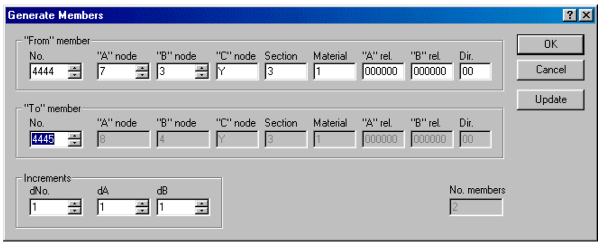
EDITING MEMBER RELEASES

The Dir. button displays the dialog box shown below for the member section orientation code. Note that this item is not relevant for sections, such as I sections, that are symmetrical about both axes.



EDITING MEMBER SECTION ORIENTATION

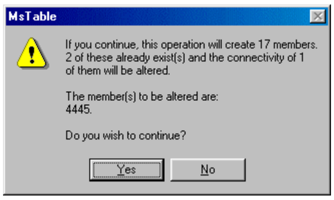
The Generate button displays the dialog box shown below.



MEMBER GENERATION

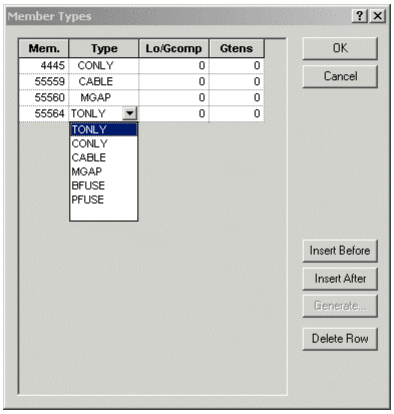
The initial “From” member is the member that was selected in the table. When you select a “From” member number that exists in the table, the member values in this dialog box will be updated automatically to those in the table.

New members are generated between the “From” and “To” members. Any existing member whose number matches a member number in the generated series will be replaced by the generated member. A message is displayed to warn you of any overwriting of members so you can decide whether to proceed or not.



MEMBER OVERWRITE WARNING

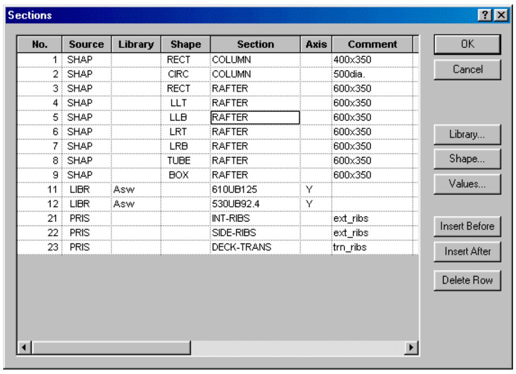
Member Types



MEMBER TYPES TABLE

To change the member type you may select from the available member types in the drop-down list box or type the code for the necessary member type as TONLY, CONLY, CABLE, MGAP, BFUSE, or PFUSE.

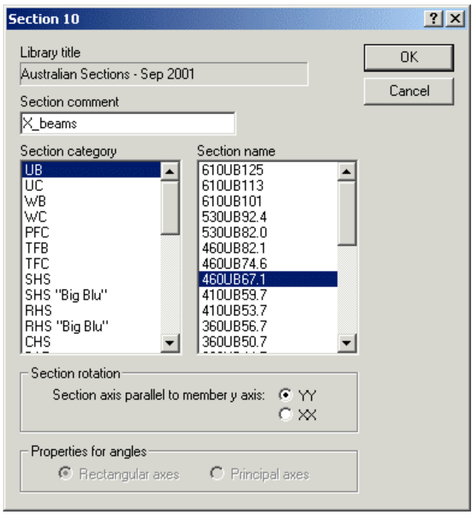
Section Properties



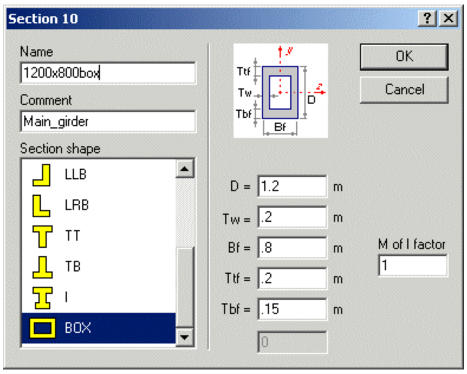
SECTION PROPERTIES TABLE

The table for section properties is shown above. The valid table entries depend on the Source field (LIBR, SHAP, or PRIS). The rightmost 6 columns of the table contain either geometric property values or shape dimensions.

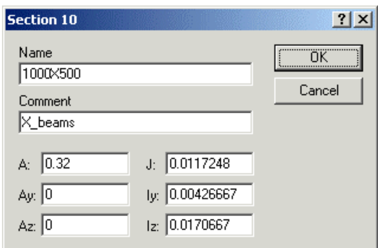
The easiest way to enter a section property is to click one of the Library, Shape, or Values buttons. Clicking these buttons displays a dialog box, as shown below, appropriate to the properties to be input.



SECTION FROM LIBRARY



SHAPE INPUT



NUMERICAL VALUES

You may paste values from the Windows clipboard into the table. For example, you can copy a range of values from a spreadsheet with Ctrl+C and paste the range directly into the table at the appropriate location with Ctrl+V. Only limited validation is performed in this table but more extensive checks are performed when you finally exit Table Input.

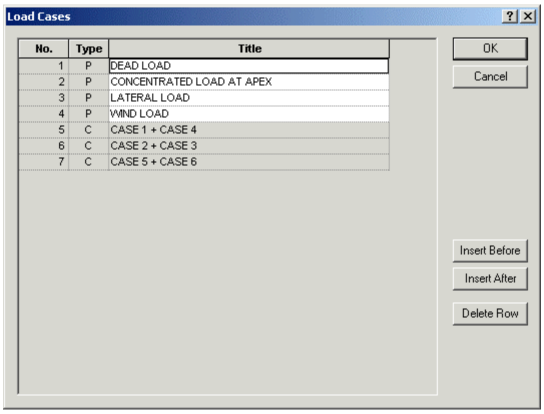
Load Case Input

The tree view shows “Primary Load Cases” and “Combination Load Cases” as top-level headings. Existing load cases may be expanded to show all load data.

Double-clicking any item in the tree that has children (as shown by the plus sign in the box) will simply expand or contract the list of children. Double-clicking an item without children will display a table so you may enter entities of the selected type.

When there are no primary load cases, double-clicking on “Primary Load Cases” will display the dialog box for entering load case titles. If primary load cases exist you may display their titles by right-clicking on “Primary Load Cases” and selecting the **Show Load Case Titles** command on the context menu. Display titles of combination load cases similarly.

The dialog box containing the load case title table is shown below.



LOAD CASE TITLES TABLE

If you displayed this dialog box from “Primary Load Cases” you may not enter data for combination load cases. Similarly, if you displayed the dialog box from “Combination Load Cases” you cannot enter data for primary load cases.

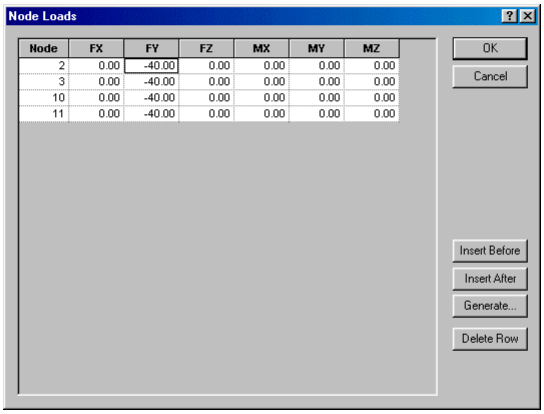
Load case numbers and load case types cannot be changed. *Deleting a row in this table deletes the corresponding load case.*

Load Types

Below is a list of all the load types that may be input with Table Input. Those shown bold are documented in detail.

- GRAV load
- **Node loads**
- Node temperatures
- Prescribed displacements
- **Member loads**
- Member temperatures
- Case combinations

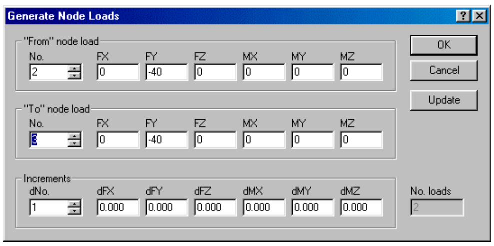
Node Loads



NODE LOADS TABLE

This dialog box contains the table of node loads for the selected load case.

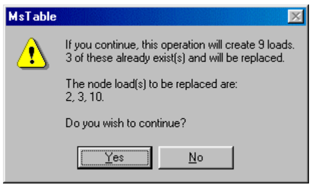
The Generate button displays the dialog box shown below.



NODE LOAD GENERATION

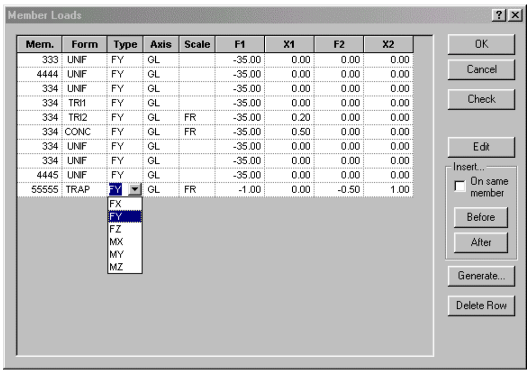
The initial “From” node is the node that was selected in the table. When you select a “From” or “To” node number that exists in the table, the load values will be updated automatically to those in the table.

New node loads are generated by interpolating between the respective load components at the “From” and “To” nodes. Any existing load at a node whose number matches a node number in the generated series will be replaced by the generated node load. A message is displayed to warn you of any overwriting of node loads so you can decide whether to proceed.



NODE LOAD OVERWRITE WARNING

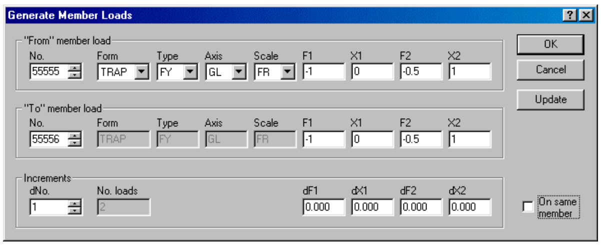
Member Loads



MEMBER LOADS TABLE

This dialog box contains the table of member loads for the selected load case. The Edit button displays the dialog box shown below so you may conveniently inspect or change the selected member load.

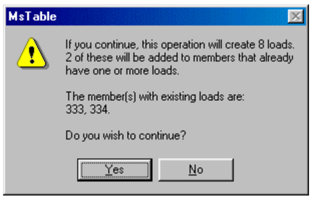
The Generate button displays the dialog box shown below.



MEMBER LOAD GENERATION

The initial "From" member is the member that was selected in the table. When you select a "From" member number that exists in the table, the corresponding load values will be updated automatically to those for the first load on that member in the table.

New member loads are generated between the "From" and "To" members. Load parameters are interpolated between the values for the "From" and "To" members. Existing member loads are not affected by generation. Each generated member load is added to the selected load case, whether or not there are existing loads on the member. A message is displayed to warn you when you are about to add loads to members that already have one or more member loads, so you can decide whether to proceed.



MEMBER LOAD OVERWRITE WARNING

9:Archive File Input

The *archive file* is a text file that describes the structure, loading, and design data. It is a convenient format for storage and transmission of job data.

Archive File Input is not a primary input method, but the expanded format, where every entity in the structure is explicitly defined, does make it useful in some situations. An archive file is created for an existing job with the **File > Export** command. It can be edited by selecting the **File > Edit File** command and it can be read into Microstran, displacing the current job, with the **File > Import** command.

The archive file is also used by third-party programs to interface to Microstran.

The archive file comprises these blocks:

1. The **Parameter Block** contains title information and specifies the type of structure.
2. The **Structure Block** contains node, member, section, and material data for the structure.
3. The **Load Block** (optional) describes the applied loading.
4. The **Design Block** (optional) contains data required for design of steel or reinforced concrete members.
5. The **Set Block** (optional) contains information about any specified sets.

The statements and groups of statements in each block are described in this chapter. You may refer to the sample archive files distributed with Microstran (one is listed in this chapter) to assist in interpreting the information that follows.

Syntax

The following rules apply to archive files:

- Items must be separated by one or more blank spaces (not commas).
- Numbers used for real values may be entered in integer, decimal, or exponential format (e.g. 1234 or 1234. or 1.234E3).

Parameter Block

Title Statements

```
* title1
* title2
```

where:

title1	First line of title (up to 60 characters of text).
title2	Second line of title (up to 60 characters of text).

Designer's Notes Group

```
* note
```

where:

note	Line of descriptive information for the job (up to 60 characters of text).

As many lines as required, each with an asterisk in the first column and a blank in the second.

Units Statement

```
UNIT  nu  ul  uf  um  ut  fl  ff  fm  ft
```

where:

nu	Unit set number. Valid unit set numbers are 1 through 6 as shown in the table below.
ul	Name of length unit.
uf	Name of force unit.
um	Name of mass unit.
ut	Name of temperature unit.
fl	Conversion factor – number of length units in 1 meter.
ff	Conversion factor – number of force units in 1 kilonewton.
fm	Conversion factor – number of mass units in 1 tonne.
ft	Conversion factor – number of temperature units in 1°C.

The unit names ul, uf, um, and ut and the conversion factors fl, ff, fm, and ft are required only for user-defined units, i.e. when the unit set number nu is 6. If the UNIT statement is omitted, units of meters, kilonewtons, tonnes, and °C will be assumed. For more information, see "Units" on page 80.

Structure Type Statement

```
TYPE  nt
```

where:

nt	Structure type. Valid types are 1 through 5.

For more information, see "Structure Type" on page 81.

Vertical Axis Statement

```
VERT  nv
```

where:

nv	Number of the global axis which is vertical in plotted views (1=X, 2=Y, 3=Z). The vertical axis may be determined by the structure type.

Structure Block

Node Coordinates Group

```
NODE  n  x  y  z  r
```

where:

n	Node number.
x y z	Coordinate values.
r	Restraint code (e.g. 111110).

For more information, see "Node Coordinates" on page 82 and "Fixed Supports (Node Restraints)" on page 87.

Spring Supports Group

```
SPRN  ns  kx  ky  kz  kmx  kmy  kmz
```

where:

ns	Node number where spring exists.
kx ky ..	Spring constants.

A spring is defined for each DOF at the nominated node for which a non-zero spring constant is specified. For more information, see "Spring Supports" on page 88.

Master-Slave Constraints Group

```
SLAV  n  n1  n2  n3  n4  n5  n6
```

where:

n	Node number.
n1 n2 ..	Number of master node for the particular DOF of the listed node – zero if no master node.

For more information, see "Master-Slave Constraints" on page 96. This data group is applicable only when the Master-Slave Constraints option is available.

Member Group

```
MEMB  m  na  nb  nc  ns  nm  mra  mrb  dir
```

where:

m	Member number.
na	End “A” node number.
nb	End “B” node number.
nc	Reference node/axis.
ns	Section number.
nm	Material number.
mra	End “A” release code.
mrb	End “B” release code.
dir	Cross-section direction code.

For more information, see “Member Definition” on page 83.

Member Type Group

```
MTYP  m  mt  Lo/Gcomp Gtens
```

where:

m	Member number.
mt	Member type – TONLY, CONLY, CABLE, MGAP, BFUSE, or PFUSE.
Lo/Gcomp	Unstrained length for cables only (if zero, chord length will be used), compression gap for gap member, maximum compression force for fuse member.
Gtens	Tension gap for gap member, maximum tension force for fuse member.

Do not include ordinary members. For more information, see “Tension-Only & Compression-Only Members” on page 97, “Cable Members” on page 98, and “Gap & Fuse Members” on page 99.

Member Modification Group

```
MOFF  m  axes  xa  ya  za  xb  yb  zb
```

where:

m	Member number.
axes	Axes in which the offsets are specified – GL for global, LO for local.
xa ya ..	Offset from node “A” to end “A” of member.
xb yb ..	Offset from node “B” to end “B” of member.

The MOFF line allows specification of rigid offsets at the ends of members. For more information, see “Rigid Member Offsets” on page 94.

```
MSPR  m  krya  krza  kryb  krzb
```

or

```
MSPR  m  kxa  krya  krza  kxb  kryb  krzb
```

where:

m	Member number.
kxa	Axial stiffness along member x axis of spring at end "A".
krya	Stiffness about member y axis of spring at end "A".
krza	Stiffness about member z axis of spring at end "A".
kxb	Axial stiffness along member x axis of spring at end "B".
kryb	Stiffness about member y axis of spring at end "B".
krzb	Stiffness about member z axis of spring at end "B".

The MSPR line allows specification of semi-rigid connections between the rigid offset and the elastic portion of the member. For more information, "Semi-Rigid Connections" on page 95.

Sections Properties Group

This data group is used to specify the geometric properties with respect to the principal axes of the member cross-sections. For more information, see "Section Properties" on page 88. The section properties for each section number may be defined by one of the input methods below:

Selection from Library

```
PROP  np  LIBR  lib_name  sect_name  axis  comment
          ax  ay  az  j  iy  iz
```

where:

LIBR	Keyword.
lib_name	Name of library – up to 10 characters.
sect_name	Name of section – up to 15 characters.
axis	Cross-section axis (either X or Y) that is coincident with the member y axis.
comment	Comment – up to 20 characters.
ax ay ..	Section geometric property values.

Note: When the nominated library file is available, Microstran takes section property and section outline data directly from the library, ignoring the values on the second line of the PROP record. If the library is not available, however, the geometric property values are used. In this case, the associated members cannot be rendered, nor checked to a steel design code.

Shape Input

```
PROP  np  SHAP  shape_code sect_name  comment
          d1  d2...
```

where:

np	Section number.
SHAP	Keyword.
shape_code	RECT, CIRC, TUBE, LLT, LRT, LLB, LRB, TT, TB, I, or BOX.
sect_name	Name of section – up to 15 characters.
comment	Comment – up to 20 characters.
d1 d2 ..	The dimensions necessary to define the shape. These are listed below for each shape.

DIMENSIONS REQUIRED TO DEFINE SHAPES

	RECT	CIRC	TUBE	L[1]	T[2]	I	BOX
d1	D	OD	OD	D	D	D	D
d2	B	-	ID	Tw	Tw	Tw	Tw
d3	-	-	-	Bf	Bf	Btf	Bf
d4	-	-	-	Tf	Tf	Ttf	Ttf
d5	-	-	-	-	-	Bbf	Tbf
d6	-	-	-	-	-	Tbf	-

[1] Applies to LLT, LRT, LLB, and LRB sections.
[2] Applies to TT and TB sections.

Property Value Input

```
PROP  np  PRIS  sect_name  comment
          ax  ay  az  j  iy  iz
```

where:

np	Section number.
PRIS	Keyword.
sect_name	Name of section – up to 15 characters.
comment	Comment – up to 20 characters.
ax ay ..	Section geometric property values.

Materials Group

```
MATL  nm  e  pr  dens  alpha
```

where:

nm	Material number.
e	Young’s modulus.
pr	Poisson’s ratio.
dens	Mass density.
alpha	Coefficient of thermal expansion.

This data group is used to specify material properties. For more information, see “Material Properties” on page 92.

Node Mass Group

```
NMAS  n  m
```

where:

n	Node number.
m	Mass.

For more information, see “Node Mass” on page 93.

Load Block

Each load case commences with a CASE statement and is terminated by either the CASE statement for the next load case, or an END statement.

Case Statement

```
CASE  lcn  desc
```

where:

lcn	Load case number.
desc	Description or load case title, up to 50 characters.

Several different load types may be included in each load case. These are set out below.

Acceleration Group

```
GRAV  gx  gy  gz
```

where:

gx	Acceleration in global X direction.
gy	Acceleration in global Y direction.
gz	Acceleration in global Z direction.

For more information, see “Acceleration Loads (GRAV)” on page 102.

Node Loads Group

```
NDLD  n  fx  fy  fz  mx  my  mz
```

where:

n	Node number.
fx fy ..	Node force values.
mx my ..	Node moment values.

For more information, see “Node Loads (NDLD)” on page 102.

Member Loads Group

```
MBLD  m  form  type  axis  scale  f1 x1  f2 x2
```

or

```
MBLD  m  DIST  type  scale  f1 x1
```

where:

m	Member number.
form	CONC, UNIF, TRAP, TRI1, TRI2.
type	FX, FY, FZ, MX, MY or MZ.
axis	GL or LO.
scale	LE or FR. May be omitted for UNIF and TRI1 loads.
f1	Magnitude at start of load.
x1	Distance from end “A” to start of load.
f2	Magnitude at end of load.
x2	Distance from end “A” to end of load.

Load forms are fully described in “Member Loads (MBLD)” on page 103. The number of parameters required depends on the load form (e.g. UNIF loads require f1 only).

Member distortions are defined as member loads with the load form keyword DIST and the parameters axis, f2, and x2 omitted.

Node Temperatures Group

```
TEMP  n  t
```

where:

n	Node number.
t	Node temperature differential.

For more information, see “Node Temperatures (TEMP)” on page 109.

Member Temperatures Group

```
MBLD  m  MTMP  tx  ty  tz
```

where:

m	Member number.

tx	Centroidal temperature differential.
ty	Effective thermal gradient in member y direction.
tz	Effective thermal gradient in member z direction.

For more information, see "Member Loads (MBLD)" on page 103.

Prescribed Displacements Group

```
NDIS  n  dx  dy  dz  rx  ry  rz
```

where:

n	Node number.
dx dy ..	Prescribed displacement components.
rx ry ..	Prescribed rotation components (radians).

For more information, see "Prescribed Node Displacements (NDIS)" on page 111.

Combination Load Case Group

```
COMB  lcn  f
```

where:

lcn	Load case number for a previously defined load case.
f	Factor by which load case is multiplied before adding to combination load case.

Load cases containing combinations must not contain any other load. For more information, see "Combination Load Cases (COMB)" on page 115.

END Statement

```
END
```

This statement terminates the load block. It must be present, even if there are no other statements in the load block.

Design Block

This block of the archive file stores information needed for Microstran's steel design and reinforced concrete design modules. It exists in archive files exported by Microstran only when one or more members have been initialized for steel or reinforced concrete design.

Keywords are shown in upper case and design parameters in lower case. The "&" character indicates the continuation of the SMEM, CBEM, and CCOL statements on the next line of the archive file.

Design Data Statement

```
$ design data
```

This statement marks the beginning of the design data.

Steel Design Group

```
SMEM m1 m2... CODE code DSEC sec GR gr MAXSL sl &
DMAX dmax &
OFFS off  TF tfc  BF bfc  LH lh   KX kx   KY ky
.....
.....
OFFS off  TF tfc  BF bfc  LH lh   KX kx   KY ky
```

where:

m1 m2 ..	Member numbers of all members in linked design member.
code	Design code.
sec	Section type mnemonic, e.g. "UB".
gr	Steel grade.
sl	Maximum slenderness ratio.
dmax	Maximum depth of section.
off	Offset from "A" end of member.
tfc	Top flange restraint codes. Any legitimate combination of these codes: N = no restraint, L = effective lateral restraint, E = elastic restraint, C = continuous restraint, R = rotational restraint (in plan).
bfc	Bottom flange restraint codes. Any legitimate combination of these codes: N = no restraint, L = effective lateral restraint, E = elastic restraint, C = continuous restraint, R = rotational restraint (in plan).
lh	Load height code: T = top flange, B = bottom flange, S = shear centre, D = destabilizing.
kx	Effective length factor for XX column buckling.
ky	Effective length factor for YY column buckling.

There is one SMEM statement for each member for which there is steel design data.

Reinforced Concrete Design Group

```
CBEM m1 [m2...] CODE code FC fc FSY fsy FSYV fsyv &
SHAPE shape D d BW bw [BF bf TF tf] &
DMTOP dmtop DMBTM dmbtm DSHEAR dshear &
NPLEG npleg MLTOP mltop MLBTM mlbtm  &
CTOP ctop CBTM cbtm TORQ torq
```

where:

m1 m2 ..	Member numbers of all members in linked design member.
code	Design code.
fc	Characteristic strength of concrete.
fsy	Yield stress of main reinforcement.
fsyv	Yield stress of shear reinforcement.
shape	Section shape code.
d	Section dimension.
bw	Section dimension.
bf	Section dimension.
tf	Section dimension.
dmtop	Minimum bar diameter, top.
dmbtm	Minimum bar diameter, bottom.
dshear	Minimum bar diameter, shear reinforcement.
npleg	Number of pairs of legs in shear reinforcement.
mltop	Maximum number of layers of reinforcement, top.
mlbtm	Maximum number of layers of reinforcement, bottom.
ctop	Clear cover to top reinforcement.
cbtm	Clear cover to bottom reinforcement.
torq	Y = design for torque, N = ignore torque.

There is one CBEM statement for each RC member to be designed as a beam.

```
CCOL m1 m2... CODE code FC fc FSY fsy SHAPE shape &
{BX bx BY by NBX nbx NBY nby | DIA dia NB nb} &
[SWAY xy] DBMIN dbmin DT dt COVER cvr QONG qong
```

where:

m1 m2 ..	Member numbers of all members in linked design member.
code	Design code.
fc	Characteristic strength of concrete.
fsy	Yield stress of main reinforcement.
shape	Section shape code (RECT or CIRC).
bx	Width of rectangular column (dimension parallel to section XX axis).
by	Depth of rectangular column (dimension parallel to section YY axis).
nbx	For rectangular column, no. bars in face parallel to section XX axis.
nby	For rectangular column, no. bars in face parallel to section YY axis.
dia	Diameter of circular column section.
nb	No. bars for circular column.
xy	Sway code – X, Y, or XY if not braced in specified section direction.
dbmin	Minimum bar diameter, main bars.
dt	Minimum bar diameter, ties.
cvr	Clear cover to ties.
qong	Ratio Q/G for design.

There is one CCOL statement for each RC member to be designed as a column.

END Statement

```
END
```

This statement terminates the design block.

Set Block

This block of the archive file stores information about any defined sets.

```
SET name
 MEMBLIST memblist
 NODELIST nodelist
 SECTLIST sectlist
 MATLIST  matlist
```

where:

memblist	A list of member numbers.
nodelist	A list of node numbers.
sectlist	A list of section numbers.
matlist	A list of material numbers.

The keywords “MEMBLIST” etc. are not required when the list is empty. List values are separated by commas or hyphens, a hyphen indicating a range of values.

END Statement

```
END
```

This statement terminates the archive file.

Archive File Example

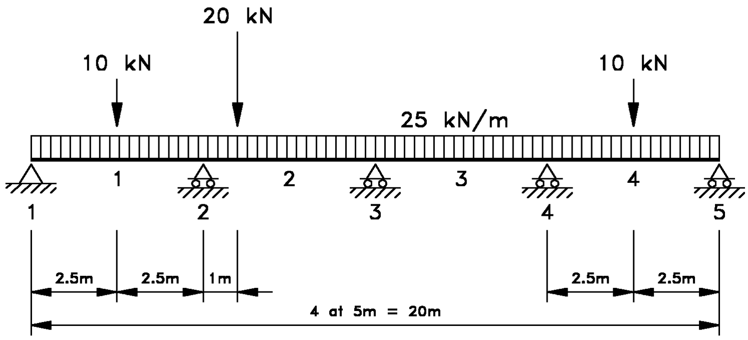
An example of an archive file is listed below. This was produced from Example 2 in Chapter 20 (see “Example 2 – Portal Frame” on page 345).

```
* PORTAL FRAME
*
*
*

VERS     5
TYPE     3
VERT     2
UNIT     1 m   kN  T   C

NODE     1     0.0000     0.0000     0.0000 111110
NODE     2     0.0000     5.0000     0.0000 001110
NODE     3    12.0000     7.4000     0.0000 001110
NODE     4    24.0000     5.0000     0.0000 001110
NODE     5    24.0000     0.0000     0.0000 111110

MEMB     1     1     2   -X      1     1    000000 000000
MEMB     2     2     3    Y      2     1    000000 000000
MEMB     3     3     4    Y      2     1    000000 000000
MEMB     4     4     5    X      1     1    000000 000000

PROP     1 LIBR Asw       610UB125        Y
      1.6000E-02  0.000      0.000     1.5600E-06 3.9300E-05 9.8600E-04
PROP     2 LIBR Asw       530UB92.4       Y
      1.1800E-02  0.000      0.000     7.7500E-07 2.3800E-05 5.5400E-04

MATL     1 2.000E+08 3.000E-01 7.850E+00 1.080E-05

CASE     1 DEAD LOAD
MBLD     2  UNIF  FY  GL           -4.000
MBLD     3  UNIF  FY  GL           -4.000

CASE     2 CONCENTRATED LOAD AT APEX
NDLD     3      0.000    -50.000      0.000      0.000      0.000      0.000

CASE     3 LATERAL LOAD
NDLD     2     50.000      0.000      0.000      0.000      0.000      0.000

CASE     4 WIND LOAD
MBLD     1  UNIF  FX  GL            3.500
MBLD     2  UNIF  FY  LO           -2.500
MBLD     3  UNIF  FY  LO            1.500
MBLD     4  UNIF  FX  GL            1.000

CASE     5 CASE 1 + CASE 4
COMB     1      1.000
COMB     4      1.000

CASE     6 CASE 2 + CASE 3
COMB     2      1.000
COMB     3      1.000

END

$ design data
SMEM 1 CODE AS4100 DSEC UB GR NORMAL MAXSL 180 DMAX 9999.
     OFFS 0.000     TF L   BF L  LH S   KX 1.000   KY 1.000
     OFFS L         TF L   BF L  LH S
SMEM 2 CODE AS4100 DSEC UB GR NORMAL MAXSL 180 DMAX 9999.
     OFFS 0.000     TF L   BF L  LH S   KX 1.000   KY 1.000
     OFFS L         TF L   BF L  LH S
SMEM 3 CODE AS4100 DSEC UB GR NORMAL MAXSL 180 DMAX 9999.
     OFFS 0.000     TF L   BF L  LH S   KX 1.000   KY 1.000
     OFFS L         TF L   BF L  LH S
SMEM 4 CODE AS4100 DSEC UB GR NORMAL MAXSL 180 DMAX 9999.
     OFFS 0.000     TF L   BF L  LH S   KX 1.000   KY 1.000
     OFFS L         TF L   BF L  LH S
END

SET RAFTERS
 MEMBLIST 2,3

SET COLUMNS
 MEMBLIST 1,4

END
```

10:Macro Language Input

General

The Macro Language Input file required to input the 4-span continuous beam shown is listed below. Comments are shown in *italics*.



MLI EXAMPLE

```
TITL1  4-Span Continuous Beam
UNITS  kN m t C
TYPE   PLANE FRAME VERT Y
NODE COORDINATES
 1 0. 0. 0.  to 5 20. 0. 0.
  $ Generate nodes at 5 m spacing.
MEMBER INCIDENCES
 1  1 2  TO  4
  $ Unit increments are assumed for member number
  $  and start and end node numbers.
REFERENCE NODES
 ALL AXIS Y
SUPPORTS
 ALL FIXED BUT FX MZ
 1 PINNED
MEMBER PROPERTIES
 ALL LIBR ASW.LIB 410UB53.7
  $ Take section properties from library file.
MATERIAL
 ALL LIBR MATL.LIB STEEL
CASE 1  Vertical loads
 MEMBER LOADS
 ALL UNIF FY GL -25.
  $ 25 kN/m throughout.
 1 4 CONC FY GL FR -10.  .5
  $ Central 10 kN load on first and last spans.
 2 3 CONC FY GL LE -20. 1.
  $ 20 kN at 1 m from start of member 2.
END
```

Microstran's Macro Language (MLI) is a special language that allows you to describe a structure and the loads in familiar terms using keywords and labels to organize the data. Node coordinates may be given in rectangular, cylindrical, or spherical systems and many powerful generation facilities are provided.

MLI is an efficient means of inputting data, particularly for regular structures. If a dimension of such a structure is changed, perhaps only one or two changes may be necessary in the MLI file.

The MLI data file is a text file with file name extension "mli", usually created and edited with a text editor, such as MsEdit, Notepad, or WordPad.

An MLI data file comprises three blocks:

1. The **Parameter Block** contains title information and specifies the type of structure.
2. The **Structure Block** contains geometry and property data for the structure.
3. The **Load Block** describes the applied loading.

The statements and groups of statements in each block are detailed in this chapter.

Syntax

The following rules apply to MLI data files:

- Items must be separated by one or more blank spaces (not commas).
- The maximum line length is 80 characters. Statements may extend over two or more lines by terminating each incomplete line with a hyphen (-).
- The semi-colon (;) may be used to separate several statements of the same kind on a single line.
- Numbers used for repeat specifications etc. must be integers.
- Numbers used for real values may be entered in integer, decimal or exponential format (e.g. 1234 or 1234. or 1.234E3).
- Keywords and labels may be abbreviated to the first four characters. Upper and lower case may be used interchangeably.
- The "$" character is used to make a comment line or add a comment at the end of a line. Any data following the $ character will be ignored by MLI.
- Where data consists of a set of labels and values (e.g. section property input) the labels may be omitted if the numeric values are entered strictly in the order shown.

Notation

The following notation is used in this manual:

- Braces indicate that at least one of the enclosed items must be selected. When the braces are shown on one line, vertical bars are used as separators (e.g. `{A | B | C}`). Neither the braces nor the vertical bars appear in the MLI statement.
- Square brackets indicate optional items (e.g. `[INC dn;]`). The brackets do not appear in the MLI statement.
- A series of dots indicates that the preceding item(s) may be repeated one or more times. The dots do not appear in the MLI statement.
- In this manual keywords and labels are shown in upper case while variables (used for text and numbers describing the structure) are shown in lower case. This convention is not required in MLI files – you may use upper and lower case interchangeably.

Lists

MLI input makes extensive use of lists to generate data. A list defines a set of numbers (e.g. node numbers) and lists may also contain the keywords `TO` and `INC` to generate part of the list.

```
n1 n2 ... ns [TO ne [INC ni;]] nj nk ...
```

or

```
ALL
```

or

```
OTHERS
```

where:

n1 n2 ..	Integer numbers.
ns	Start of generated sequence.
TO	Keyword.
ne	End of generated sequence.
INC	Keyword.
ni	Optional generation increment (defaults to 1).
nj nk ..	Integer numbers.
ALL	Keyword defining a list that includes all elements.
OTHERS	Keyword defining a list that includes all elements not already specified.

The nodes `ns` and `ne` must have been previously defined and the increment `ni` must be consistent with the difference between start and end values, i.e. the range specified must be an exact multiple of the increment. Any list may be extended over two or more lines by placing a hyphen at the end of each incomplete line.

Examples:

```
1 2 5 7 11 12 17             $ Discrete items
7 TO 16 INC 3                $ Generated list
1 2 5 7 TO 16 INC 3; 21 23   $ Composite list
```

Parameter Block

Title Statements

```
TITL1 title1
TITL2 title2
```

where:

title1	First line of title (up to 60 characters of text).
title2	Second line of title (up to 60 characters of text).

Units Statement

```
UNITS  {m }  {kN }  {t  }  {C}
       {mm}  {N  }  {kg }  {F}
       {ft}  {kip}  {lbs}
       {in}
```

The UNITS statement defines the units of length, force, mass, and temperature to be used. The set of units chosen should be consistent (e.g. meters, kilonewtons, tonnes). Values extracted from the section library and the material library will be transformed to the specified structure units. For more information refer to “Units” on page 80.

If the UNITS statement is omitted, units of meters, kilonewtons, tonnes, and °C will be assumed.

Structure Type Statement

```
TYPE PLANE {TRUSS | FRAME | GRID}
```

or

```
TYPE SPACE {TRUSS | FRAME} VERT {X | Y | Z}
```

Plane structures always lie in the XY global plane. Plane trusses and frames have the Y axis vertical while grids have the Z axis vertical. Space structures may be defined with any global axis vertical. See “Structure Type” on page 81.

The structure type and vertical axis are used to select the initial graphics view.

Structure Block

Node Coordinates Group

```
NODE COORDINATES
n1 x1 y1 z1; n2 x2 y2 z2; ... ; -
  ns xs ys ys [TO ne xe ye ze [INC dn;]]
```

where:

n1 n2 ..	Node numbers.
x1 y1 ..	Coordinate values.
ns	Number of first generated node.
xs ys zs	Coordinates of first generated node.
TO	Keyword to introduce last generated node.
ne	Number of last generated node.
xe ye ze	Coordinates of last generated node.
INC	Keyword to introduce node number increment.
dn	Node number increment for generation.

Any number of node coordinates may be defined on a single line by using a semi-colon (;) as a separator. Long lines may be broken into two or more lines by using a hyphen (-) at the end of each incomplete line. Uniformly spaced nodes may be generated by specifying the TO node and optionally, the node number increment. Nodes are generated in the current coordinate system. If the increment is not specified it is assumed to be 1.

Coordinate values specified on any statement refer to the current axis system, which will be the global rectangular system unless the statement is preceded by an axis system statement or a secondary axis system statement (see below). If a cylindrical or spherical axis system is in effect, the coordinates (x1,y1,z1) above will actually be (R,Θ,Z) or (R,Θ,Φ) coordinates, respectively. See "Node Coordinates" on page 82.

```
REPEAT [ALL] nrep INC dn dx dy dz
```

where:

nrep	Number of repeats.
INC	Keyword to introduce repeat increment values.
dn	Node number increment for each repeat.
dx dy dz	Coordinate increments for each repeat.

The REPEAT statement may appear in the node coordinates group to copy previously entered coordinate data. If the keyword ALL is present the repeat action is carried out for all the nodes input since the last REPEAT ALL command. If no REPEAT ALL command has been given the repeat action is carried out from the beginning of the node coordinate input. If ALL is omitted the repeat action is carried out only for the

nodes input by the previous line in the MLI file (lines that have been extended with a hyphen at the end are regarded as a single line by the REPEAT function).

Axis System Statement

```
{RECTANGULAR | CYLINDRICAL | SPHERICAL}
```

This statement may appear anywhere within the node coordinates group to define the coordinate system to be used for subsequent input. The coordinates of the nodes of the structure are given in two or three dimensional space. Rectangular, cylindrical, or spherical systems may be used. All coordinates following this declaration will be in the specified coordinate system.

RECTANGULAR (X,Y,Z)

Rectangular coordinates are the default coordinate system and will be used if no other system is declared. For structures wholly within the XY plane, Z coordinates may be omitted.

CYLINDRICAL (R,Θ,Z)

In cylindrical coordinates, R is the radius, Θ is the rotation in degrees about the Z axis, measured in a right-handed sense from the X axis, and Z is the height.

SPHERICAL (R,Θ,Φ)

In spherical coordinates, R is the radius, Θ (the longitude coordinate) is the rotation in degrees about the Z axis, measured in a right-handed sense from the X axis, and Φ (the latitude coordinate) is the angle measured from the XY plane, positive if the node is above the plane and negative if below it.

At the completion of node coordinate input all coordinates are transformed to the primary rectangular axis system. You may change axis systems as many times as necessary.

Secondary Axis System Statement

```
{RECTANGULAR}   ORIGIN XC x YC y ZC z AXIS  {X}   ZERO   {X}
{CYLINDRICAL}                               {Y}          {Y}
{SPHERICAL  }                               {Z}          {Z}
```

where:

ORIGIN	Keyword to introduce coordinates of origin of secondary axis system.
XC YC ZC	Labels for the origin coordinate values.
x y z	Coordinates of the new origin.
AXIS	Keyword to denote the name of the primary axis which is parallel to the secondary Z axis.
ZERO	Keyword to denote the name of the primary axis which is parallel to the secondary X axis.

X Y Z	Axis names.

Additional fields may be used with the RECTANGULAR, CYLINDRICAL, and SPHERICAL keywords to specify a secondary axis system with an origin and axis orientations which may differ from those of the primary rectangular system. If no AXIS or ZERO keyword is used, the secondary axes are parallel to the primary axes.

Examples:

```
SPHER XC 10 YC 10 ZC 10 AXIS Z ZERO X
```

This statement defines a set of spherical axes with origin at (10,10,10) in the primary system. In this case, the X, Y and Z axes of the two sets of axes are parallel.

```
CYLINDRICAL AXIS X ZERO Y
```

This statement defines a set of cylindrical axes with the same origin as the primary axes, the cylindrical Z axis parallel to the primary X axis and the cylindrical X axis parallel to the primary Y axis. This would be appropriate for the definition of the nodes of a tubular structure aligned with the primary X axis.

Coordinate Transformation Group

Translational and rotational transformations may be applied to node coordinates by using the TRANSLATE and ROTATE keywords in the node coordinates group. The transformation commands allow you to define node coordinates with the structure in the most convenient position and then re-position it as required.

```
TRANSLATE  {ALL | list}  DX dx DY dy DZ dz
```

where:

DX DY DZ	Labels for translation offset values.
dx dy dz	Distances by which nodes are moved.

The TRANSLATE statement causes the nodes of the complete structure or a part of it to be moved with respect to the global axes. All coordinates to be translated are transformed to the primary rectangular axis system before the translation is applied. Zero components of the translation do not need to be specified.

```
ROTATE  {ALL | list}  RX rx RY ry RZ rz
```

where:

RX RY RZ	Labels for rotation angles.
rx ry rz	Rotation angles (degrees).

The ROTATE statement causes the nodes of the complete structure or a part of it to be rotated about the global axes. Positive rotations cause the structure to move in a right-handed sense about the nominated axis of the primary rectangular axis system. Zero rotation angles do not need to be specified. Rotations are applied to the node coordinates in the order in which they are specified and do not need not be given in X, Y, Z order.

Note that where multiple rotations are specified, the final orientation depends on the order in which the rotations are specified.

Supports Group

```
SUPPORTS
list PINNED
list FIXED  [BUT release_list [spring_list]]
```

where:

list	List of nodes that are supports.
PINNED	Keyword which denotes listed nodes as pinned supports (i.e. translational DOF restrained, rotational DOF not restrained).
FIXED	Keyword which denotes listed nodes as fixed supports (i.e. translational and rotational DOF fixed).
release_list	List of actions to be released at support node; any set from the labels FX FY FZ MX MY MZ.
spring_list	List of label/stiffness pairs; any set of pairs from KFX/kfx KFY/kfy KFZ/kfz KMX/kmx KMY/kmy KMZ/kmz. KFX is a label introducing the stiffness value kfx; KMX is a label introducing the rotational stiffness value kmx.

Any individual degrees of freedom (DOF) may be released by listing them after the keyword BUT. However, those DOF suppressed because of the structure type cannot be released. For example, FZ cannot be released in a plane frame. See “Fixed Supports (Node Restraints)” on page 87 and “Spring Supports” on page 88.

Examples:

```
$ Roller support -
 10 FIXED BUT FX MZ
$ Support with vertical spring -
 20 FIXED BUT MZ KFY 2.3E6
$ Support with rotational spring -
 30 FIXED BUT KMZ 4.5E6
```

Master-Slave Constraints Group

```
MASTER SLAVE

list MAST DX n1 DY n2 DZ n3 RX n4 RY n5 RZ n6
```

where:

list	List of nodes having the specified constraints. These are slave nodes.
MAST	Keyword introducing constraint relationships.
DX DY ..	Labels identifying DOF for which master node follows.
n1 n2 ..	Number of master node for the particular DOF of the listed node.

Unconstrained DOF should be omitted. This data group is applicable only when Master-Slave Constraints is available. See “Master-Slave Constraints” on page 96.

Member Incidences Group

```
MEMBER INCIDENCES
m1 na1 nb1; m2 na2 nb2; ... ; -
  ms nas nbs [TO me [INC dm dna dnb;]]
```

where:

m1 m2 ..	Member numbers
na1 na2 ..	End “A” node numbers.
nb1 nb2 ..	End “B” node numbers.
ms	Number of first generated member.
nas	End “A” node number of first generated member.
nbs	End “B” node number of first generated member.
TO	Keyword to introduce last generated member number.
me	Number of last generated member.
INC	Keyword to introduce generation increment values.
dm	Member number increment for generation.
dna	End “A” node number increment for generation.
dnb	End “B” node number increment for generation.

Any number of member incidences may be defined on a single line by using a semi-colon (;) as a separator. Long lines may be broken into two or more lines by using a hyphen (-) at the end of each incomplete line. Regularly numbered members may be generated by specifying the TO member and the (optional) member number and end node increments. Any increments not specified are assumed to be 1. See “Member Definition” on page 83.

```
REPEAT [ALL] nrep INC dm dna dnb
```

where:

nrep	Number of repeats.
INC	Keyword to introduce repeat increment values.
dm	Member number increment for each repeat.
dna	End “A” increment for each repeat.
dnb	End “B” increment for each repeat.

The REPEAT statement may appear in the member group to copy previously entered member incidence data. If the keyword ALL is present the repeat action is carried out for all the members given since the last REPEAT ALL command. If no REPEAT ALL command has been given the repeat action is carried out from the beginning of the member incidence input. If ALL is omitted the repeat action is carried out only for the members input by the previous line in the MLI file (lines that have been extended with a hyphen at the end are regarded as a single line by the REPEAT function).

Member Types Group

```
MEMBER TYPES
list TYPE {TONLY | CONLY}
```

or

```
MEMBER TYPES
list TYPE CABLE Lo
```

or

```
MEMBER TYPES
list TYPE {MGAP | BFUSE | PFUSE} Gcomp Gtens
```

where:

list	List of members which are of following type.
TYPE	Keyword to introduce member type.
TONLY	Keyword which denotes listed members as tension-only.
CONLY	Keyword denotes listed members as compression-only.
CABLE	Keyword denotes listed members as cables.
MGAP	Keyword denotes listed members as gap members.
BFUSE	Keyword denotes listed members as brittle fuse members.
PFUSE	Keyword denotes listed members as plastic fuse members.
Lo	Unstrained length of cable (0 = chord length).
Gcomp	Compression gap for gap members, max. compression force for fuse members.
Gtens	Tension gap for gap members, max. tension force for fuse members.

All these member types require non-linear analysis.

Reference Node/Axis Group

```
REFERENCE NODE
list NODE n
```

or

```
REFERENCE NODE
list AXIS axis
```

where:

list	List of members.
NODE	Keyword to introduce number of reference node.
n	Number of reference node “C” used to define member xy plane.
AXIS	Keyword to introduce reference axis.
axis	Name of global axis (any one of X Y Z -X -Y -Z) used to define the member xy plane.

The position of the reference or “C” node (together with the member nodes “A” and “B”) is used to define a plane that contains the member *y*

axis. The transverse member axes *y* and *z* are the principal axes of the member cross-section. The positive *y* axis lies on the same side of the member as the "C" node. The coordinates of the reference node must have been defined. See "Member Definition" on page 83.

If a reference axis is specified the member *xy* plane is parallel to the nominated global axis and the member *y* axis points towards that axis. A default reference axis will be chosen for any member that is not included on a reference node/axis list. The rules for determining which reference axis will be used by default are set out in Chapter 5 – "Structure & Load Data".

Member Releases Group

```
MEMBER RELEASES
list ENDA lista ENDB listb
```

where:

list	List of members with specified releases.
ENDA	Keyword to introduce releases at end "A" of member.
lista	List of released actions – any labels from the set FX FY FZ MX MY MZ.
ENDB	Keyword to introduce releases at end "B" of member.
listb	List of released actions – any labels from the set FX FY FZ MX MY MZ.

Force or moment releases may be specified at the ends of members. Any force may be released provided that the member remains stable and attached to the structure. See "Member Definition" on page 83.

Member Modification Group

```
RIGID OFFSETS
list OFFSET XA xa YA ya ZA za XB xb YB yb ZB zb
```

where:

list	List of members with specified modification.
OFFSET	Keyword to introduce member offset data.
XA YA ..	Keywords to introduce offset values.
xa ya ..	Offset in global axis directions from node "A" to end "A" of member.
XB YB ..	Keywords introducing offset values.
xb yb ..	Offset in global axis directions from node "B" to end "B" of member.

```
SEMI RIGID
list SPRINGS KXA kxa KRYA krya KRZA krza -
             KXB kxb KRYB kryb KRZB krzb
```

where:

list	List of members with specified modification.
SPRINGS	Keyword introducing semi-rigid connection data.
KXA	Keyword introducing stiffness in member x axis direction of spring at “A”.
kxa	Stiffness value.
KRYA	Keyword introducing stiffness about member y axis of spring at “A”.
krya	Stiffness value.
KRZA	Keyword introducing stiffness about member z axis of spring at “A”.
krza	Stiffness value.
KXB	Keyword introducing stiffness in member x axis direction of spring at “B”.
kxb	Stiffness value.
KRYB	Keyword introducing stiffness about member y axis of spring at “B”.
kryb	Stiffness value.
KRZB	Keyword introducing stiffness about member z axis of spring at “B”.
krzb	Stiffness value.

This data group allows the specification of rigid offsets at the ends of members and semi-rigid connections between the rigid offset and the elastic portion of the member. See “Rigid Member Offsets” on page 94 and “Semi-Rigid Connections” on page 95.

Member Properties Group

```
MEMBER PROPERTIES
list PRIS AX ax AY ay AZ az IX ix IY iy IZ iz
```

or

```
list LIBR lib_name sect_name
```

where:

list	List of members with specified section properties.
PRIS	Keyword.
AX AY ..	Labels introducing section property value.
ax ay ..	Section geometric property values. The geometric properties about the principal axes of the member cross-section are specified. Properties must be defined for all members. Any omitted properties are set to zero. If the keywords AX AY etc. are not used, properties must be given in the specified order.
LIBR	Keyword.
lib_name	Name of library file. The file name including extension must

	not exceed 16 characters. The file name extension ".lib" will be added if no extension is specified.
sect_name	Name of section. Abbreviations may be used; the first section whose name matches the abbreviation will be selected.

See "Section Properties" on page 88.

Materials Group

```
MATERIALS
list PROP E e PR pr DENS dens ALPHA alpha
```

or

```
list LIBR lib_name matl_name
```

where:

list	List of members with specified material properties.
PROP	Keyword.
E	Label introducing Young's modulus value.
e	Young's modulus.
PR	Label introducing Poisson's ratio value.
pr	Poisson's ratio.
DENS	Label introducing mass density value.
dens	Mass density.
ALPHA	Label introducing coefficient of thermal expansion.
alpha	Coefficient of thermal expansion.
lib_name	Name of library file. The file name including extension must not exceed 16 characters. The file name extension ".lib" will be added if no extension is specified.
matl_name	Name of material. Abbreviations may be used; the first material in the library whose name matches the abbreviation will be selected.

This data group is used to specify material properties of members. Materials must be defined for all members. If the keywords E, PR, DENS, and ALPHA are not used in the PROP statement, the property values must be given in the above order. Any omitted properties are set to zero. See "Material Properties" on page 92.

Load Block

Each load case commences with a CASE statement and is terminated either by the CASE statement for the next load case or by the END statement that marks the end of the MLI file.

Case Statement

```
CASE lcn desc
```

where:

lcn	Load case number.
desc	Description or load case title, up to 50 characters.

Several different load types may be included in each load case. These are set out below.

Acceleration Group

```
GRAVITY LOADS
GX gx GY gy GZ gz
```

where:

gx	Acceleration in global X direction.
gy	Acceleration in global Y direction.
gz	Acceleration in global Z direction.

See “Acceleration Loads (GRAV)” on page 102.

Node Loads Group

```
NODE LOADS
list FX fx FY fy FZ fz MX mx MY my MZ mx
```

where:

list	List of nodes to which the loads are applied.
FX FY ..	Labels introducing fx fy etc.
fx fy ..	Node force values.
MX MY ..	Labels introducing mx my etc.
mx my ..	Node moment values.

See “Node Loads (NDLD)” on page 102.

Examples:

```
NODE LOADS
1 2 3 TO 9 INC 2 FX -1.2
OTHERS FZ 5.1
```

Member Loads Group

```
MEMBER LOADS
list form type axis scale F1 f1 [X1 x1 [F2 f2 X2 x2]]
```

where:

list	List of members to which the load is applied.
form	CONC, UNIF, TRAP, TRI1 or TRI2.
type	FX, FY, FZ, MX, MY, or MZ.
axis	GL or LO.
scale	LE or FR. May be omitted for UNIF and TRI1 loads.
F1 F2	Keywords introducing f1 f2.
f1	Magnitude at start of load.
f2	Magnitude at end of load.
X1 X2	Keywords introducing x1 x2.
x1	Distance from end “A” to start of load.
x2	Distance from end “A” to end of load.

See “Member Loads (MBLD)” on page 103.

Depending on the load form, some of the numerical values shown in parentheses may be omitted (e.g. UNIF loads require f1 only). Keywords for numerical parameters may be omitted if load values are given in the above order.

Member Distortions

Member distortions may be defined as member loads with the load form keyword DIST. See “Member Distortions” on page 107.

```
list DIST {FX | ... | MZ} {LE | FR} F1 f1 X1 x1
```

Examples:

```
MEMB LOADS
101 TO 199 INC 2 UNIF FY GL -2.1
121 122 123 TRAP FY LO FR F1 1.1 X1 0. F2 2.1 X2 1
145 CONC FY GL FR -10.5 .5
```

Node Temperatures Group

```
NODE TEMPERATURES
list TEMP t
```

where:

list	List of nodes to which the specified temperature differential applies.
t	Node temperature differential.

See “Node Temperatures (TEMP)” on page 109.

Member Temperatures Group

```
MEMBER TEMPERATURES
list TEMP  TX tx  TY ty  TZ tz
```

where:

list	List of members to which the specified temperature applies.
tx	Centroidal temperature differential.
ty	Effective temperature gradient in member y direction.
tz	Effective temperature gradient in member z direction.

See “Member Temperatures (MTMP)” on page 110.

Prescribed Displacements Group

```
NODE DISPLACEMENTS
list FX dx FY dy FZ dx MX rx MY ry MZ rz
```

where:

list	List of nodes at which the displacements are prescribed.
FX FY ..	Labels introducing dx dy etc.
dx dy ..	Prescribed displacement values.
MX MY ..	Labels introducing rx ry etc.
rx ry ..	Prescribed rotation values (radians).

See “Prescribed Node Displacements (NDIS)” on page 111.

Combination Load Case Group

```
COMBINATION
lc1 f1; lc2 f2; ... ; lcn fn
```

where:

lc1 lc2 ..	Load case number for a previously defined load case.
f1 f2 ..	Factor by which load case is multiplied before adding to combination load case.

The new load case is then:

$lc = f1 \times lc1 + f2 \times lc2 + \ldots + fn \times lcn$

Load cases containing combinations must not contain any other load types and must be defined after all primary load cases have been specified. See “Combination Load Cases (COMB)” on page 115.

END Statement

```
END
```

The last statement in an MLI file contains the keyword `END`.

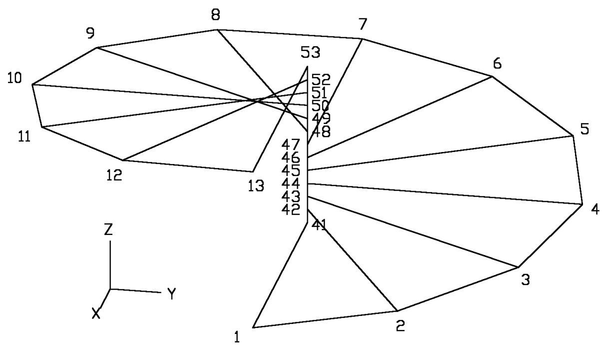
MLI Errors

If any errors are found in the MLI file, either syntax errors or missing data, the MLI data and appropriate error messages will be written to the error log. After detecting an error the program proceeds with syntax checking but if more than ten errors are found checking will cease and the remainder of the input file will be listed in the error file. For syntax errors, the data line that caused the error will be listed with the error message.

MLI Example

The MLI file listed below generates a spiral staircase structure as shown in the diagram.

```
TITL1  Spiral Stair
TITL2  MLI Example
UNITS kN m t C
TYPE SPACE FRAME VERT Z
NODE COORDINATES
 CYLIND
  1  4.  0.  0.                $ Start of spiral
  REP 12 INC  1  0. 30. .2     $ Generate spiral
  41 0. 0. 0. TO 53 0. 0. 2.4  $ Spine  (INC 1)
SUPPORTS
 1 13 FIXED
 41 53 PINNED
MEMBER INCIDENCES
 1  1  2  TO 12                $ Spiral  (INC 1 1 1)
 41 41 42 TO 52                $ Spine   (INC 1 1 1)
 101 1 41 TO 113               $ Treads  (INC 1 1 1)
REFERENCE NODE
 1 TO 12 ; 101 TO 113 AXIS Z
 OTHERS AXIS X
MEMBER PROPERTIES
 1 TO 12 LIBR ASW.LIB 250x150x6.0RHS    $ Spiral
 41 TO 52 LIBR ASW.LIB 219.1x6.4CHS     $ Spine
 OTHERS LIBR ASW.LIB 150x100x6.0RHS     $ Treads
MATERIAL
 ALL LIBR MATL.LIB STEEL
CASE 1  Dead Load
 GRAVITY LOADS
  GZ -9.8
CASE 2  Tread Loads
 NODE LOADS
  1 TO 12 FZ -2.1
 MEMB LOADS
  1 TO 12 UNIF FZ GL -1.1
CASE 3  Case 1 + Case 2
 COMB
  1  1. ; 2 1.2
END
```

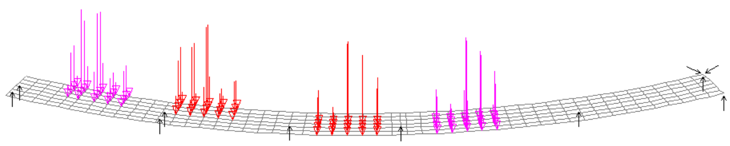


MLI EXAMPLE – SPIRAL STAIRCASE

11:Moving Load Generator

General

This optional module may be applied to any Microstran model and is particularly useful for generating service loads on bridge structures. Options are available to apply loads automatically to continuous beams or grillages in the Microstran model.



DECK1 EXAMPLE – T44 TRUCK LOADINGS ON CURVED BRIDGE DECK

Available load types include fixed distributed and concentrated loads, area loads, truck loadings (for standard trucks and user-defined trucks), lane loads, longitudinal loads, and lateral line loads. Multiple load cases can be specified and new loads can be generated automatically to represent the movement of a specified load pattern across the structure.

Assumptions

- Grillage models must have the global Z axis vertical (all loads are generated as global Z loads).
- The direction of the moving load is generally along the global X axis for loads on beams and along a lane for loads on grillages. The specification of lanes is discussed below.

Generating the Loads

To generate the required loads, select the **Loads > Moving Loads > Beam** command or the **Loads > Moving Loads > Grillage** command. You will then be prompted to select the load-carrying members of the beam or grillage. Select the members, right-click, and confirm the selection. Making a *set* of the required members will facilitate their rapid selection.

Loads are specified in a BBL file for beams and an HBL file for grillages. The BBL file is edited with the text editor while the HBL file is managed by the graphical interface and is usually never seen.

The BBL file is usually created from a template file, which is created automatically if the BBL file does not already exist. The text file is a free-format file with one or more spaces used to separate data items. The maximum line length is 80 characters.

The formats used for beams and grillage files differ and are described in the following pages. Generally the file consists of:

- Definition of the areas or lanes to be loaded (grillages only).
- Definition of special loads.
- Definition of base load cases of various types.
- Specification that generates load cases by incrementally positioning the base load cases.

Truck Types

A number of standard truck types are built into the program and may be referenced by name. Standard loadings are available for the following trucks:

- **T44F, T44B** – AUSTROADS T44 truck loading. The suffixes **F** and **B** are used to position the truck facing forwards or backwards with respect to the positive lane direction.
- **T54F, T54B** – Semi-trailer/B-double truck. The suffixes **F** and **B** are used to position the truck facing forwards or backwards with respect to the positive lane direction. The axle spacing of the semi-trailer may be varied from 3 to 8 meters; the overall length of the truck is 21 meters..
- **M1600F, M1600B** – New AUSTROADS moving loads model. The suffixes **F** and **B** are used to position the truck facing forwards or backwards with respect to the positive lane direction.
- **HLP300, HLP400** – AUSTROADS Heavy Load Platform loading. The variable spacing is either 0, or in the range from 6 to 15 meters..
- **HBnm** – BS 5400 HB loading. **nm** is the number of units of HB loading (10 kN/axle) to be applied. The variable spacing is checked for compliance with the values specified in BS 5400.
- **SVnnuuuD** – Malaysian JKR SV loading. **nn** is the number of axles in the SV trailer, **uuu** is the number of 10 kN units of loading in each of the trailer axes, and **D** is the direction the truck, either “F” or “B”. For example, SV20007F represents a forward-facing SV truck with a twenty axle trailer with each axle of the trailer having a weight of 70 kN.

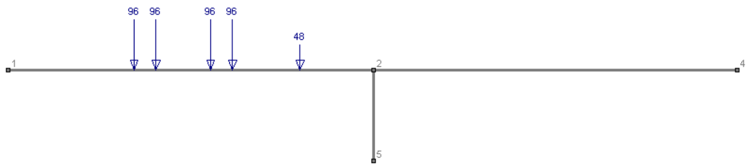
You may define trucks other than the standard trucks by making up a group of point loads using the TRUCK command. Where other types of load, such as knife-edge or area loads, are involved they can be

combined as a base load case and then used to generate moving loads in subsequent load cases.

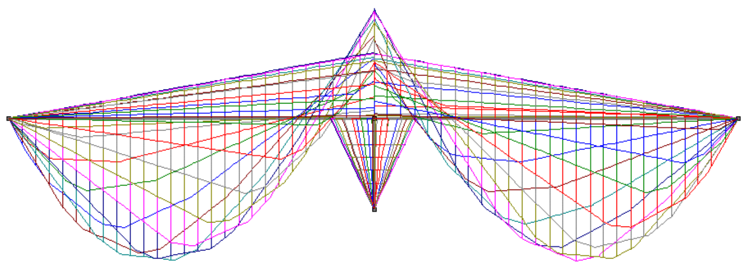
The reference point for positioning standard truck loadings on grillages is the central point load for T44, and the centroid of the load group for HLP and HB loadings. *The reference point for standard trucks on beams is the axle furthest from the start of the beam*. The reference point for user-defined trucks is the origin of the truck coordinates.

Example

The diagrams below illustrate the example, Beam1, which is included in the examples folder when you install Microstran. The first diagram shows one position of a T44F truck on a 2-span bridge. The second shows the bending moment diagram for all generated load cases, each corresponding to a different location of the truck. Microstran permits the display and reporting of member force envelopes, which are convenient when dealing with moving loads.



BEAM1 EXAMPLE – T44F TRUCK



BEAM1 EXAMPLE – BM DIAGRAM FOR ALL TRUCK POSITIONS

Moving Loads – Beams

Moving loads are applied to a series of connected beams, which may lie on a curve having both vertical and horizontal curvature. Load positions on beams are determined from the start of the beam, which is the end of the selected sequence of members having the smallest X coordinate.

The load file is a text file named Job.BBL, where “Job” represents the job name. It consists of one or more CASE blocks followed by an END statement. Comments may be included after the “$” character.

```
CASE  n  title
  [OUTPUT]
  [Load type statements]
  [OFFSET statements]
CASE...
.....
.....
END
```

where:

CASE	Keyword marking start of new load case.
N	Load case number.
Title	Load case title, up to 50 characters.
OUTPUT	Keyword that specifies output of this load case to the Microstran job. This allows a logical arrangement of the load data in the BBL file with full control over the number of load cases to be analysed. For example, if an assessment has to be made of the effects of trucks in different spans of the bridge, each truck could be assigned to a separate load case but output could be restricted to those combinations with individual cases having a particular offset.

The load type and OFFSET statements are described below.

Load Types – Beams

Fixed Distributed Load – Beams:

This is a distributed load of constant or varying magnitude. It is fixed in position and cannot be moved as part of an OFFSET load case.

```
WUDL   distance  length  w1  w2
```

where:

Distance	The distance from the start of the beam to the nearest end of the load.
Length	The length of the load.
w1	The load intensity at the start of the WUDL load.
w2	The load intensity at the end of the WUDL load.

Fixed Concentrated Load – Beams:

This is a concentrated load. It is fixed in position and cannot be moved as part of an OFFSET load case.

```
CONC   distance  load
```

where:

distance	The distance from the start of the beam to the load.
load	The value of the CONC load.

Line Load – Beams:

This is a distributed load of constant magnitude. It may be moved along the beam as part of an OFFSET load case.

```
LINEL  distance  length  load
```

where:

distance	The distance from the start of the beam to the nearest end of the load.
length	Length of line load.
load	Load per unit length.

Point Load – Beams:

This is a point load that may be moved along the beam as part of an OFFSET load case.

```
POINT  distance  load
```

where:

distance	The distance from the start of the beam to the load.
load	The value of the POINT load.

User-Defined Trucks – Beams

User-defined trucks or other point load groups may be defined as a table of coordinates and load values. Each user-defined truck has its own coordinate system, used to locate its axles.

```
TRUCK name
x  P   .....
.....
END
```

where:

name	Character string of one to eight characters used to identify the load group.
x P	Coordinate and load value. A number of (x P) groups may be entered on each line. The first x value is conventionally zero.

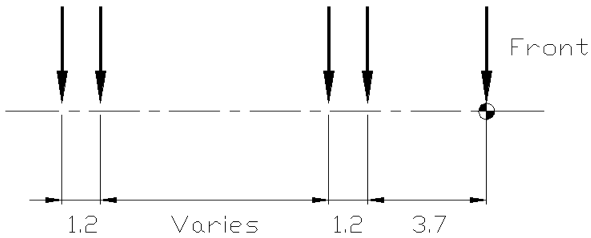
Truck Loading – Beams:

The group loading can be used for standard trucks or user-defined trucks that have been previously defined with a TRUCK statement. A group load may be moved along the beam as part of an OFFSET load case.

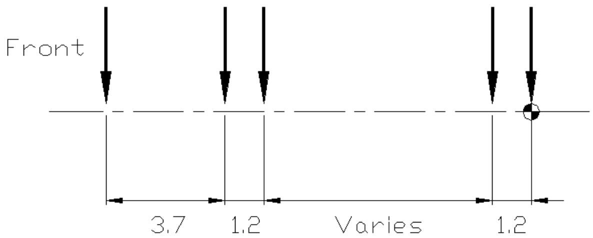
```
GROUP distance  truck  var_space  factor
```

where:

distance	The distance from the start of the beam to the load origin. The load origin for a standard truck is the axle furthest from the start of the beam.
truck	Truck name of a standard or user-defined truck.
var_space	Standard truck loading parameter. Ignored if not applicable (e.g. user-defined truck).
factor	Factor applied to truck loads (e.g. impact factor).



T44F TRUCK LOADING



T44B TRUCK LOADING

Offset Load Cases – Beams

New load cases may be generated by offsetting previously defined load cases with the OFFSET statement. Any load case containing offset cases must not itself be offset. There may be more than one OFFSET statement in each CASE block. Each generates at least one load case, whose number is the number on the preceding CASE statement. A series of additional load cases is generated according to the parameters, noff and lc_inc. These parameters are the same for all OFFSET statements in any CASE block and they may be omitted for all but the first OFFSET statement in each CASE block.

```
OFFSET  lcase  start  inc [ noff  lc_inc ]  [F  f]
```

where:

lcase	Number of previously defined load case to be offset.
start	Offset distance of first load case from base load case, lcase.
inc	Incremental distance along beam for each additional generated load case. May be zero if noff is zero or if all generated loads are at the same location.
noff	Number of offset cases to be generated, apart from the first. Zero if only one new load case is required.
lc_inc	Load case number increment for additional generated load cases. May be zero if noff is zero.
F	Keyword to specify factor for the offset load case.
f	Factor applied to generated loads.

No other load type may be included in a CASE block containing an OFFSET statement. WUDL and CONC loads are fixed and will not be offset if included in a load case referred to by an OFFSET statement.

Example – BBL File

```
2-Span Bridge as Beam

CASE 1  Truck at start of bridge
 $OUTPUT
 GROUP  0 T44F 3.0 1.0   $ distance  truck  var_space  factor

CASE 2 Fixed UDL on span 2
 $OUTPUT
 WUDL 20  40  -25 -25    $ distance  length  w1  w2

CASE 3 Fixed conc. load on span 1
 $OUTPUT
 CONC  15  -25           $ distance  load

CASE 4 Movable UDL at start of bridge
 $OUTPUT
 LINEL 0 10 -30          $ distance  length  load

CASE 100 Move truck along span 1
 $ Note fixed UDL on span 2 for all generated cases
 $OUTPUT
 OFFSET 1 0.0  1.0  20 1 $ lcase  start  inc  noff  lc_inc
 OFFSET 2 0.0  0.0       $ lcase  start  inc

CASE 150 Move truck along bridge
 OUTPUT
 OFFSET 1   0.0 2.0 24 1 $ lcase  start  inc  noff  lc_inc

CASE 200 Move UDL along bridge
 $OUTPUT
 OFFSET 4 -10.0 2.0 24 1 $ lcase  start  inc  noff  lc_inc

END
```

On installation of Microstran, this example is included in the examples folder as Beam1. The diagrams on page 201 show the structure and the bending moment diagram resulting from the CASE 150 block.

Moving Loads – Grillages

Moving loads may be applied to a grillage of members, provided that no members of the grillage cross without intersecting.

Loads are defined in a text file named Job.HBL, where "Job" represents the job name. Comments may be included after the "$" character.

Grillage loadings are located by reference to lanes or areas.

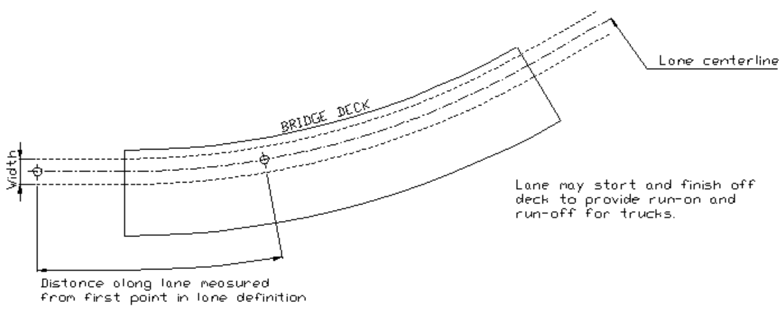
Lane Definition

A lane must be defined for all types of loading except UDL and AREA loads. Lanes are paths along which loads may be moved. Each is defined as a sequence of straight and curved segments by inputting centre-line coordinates, together with widths and radii of curvature. There is one LANE block for each lane, followed by an END statement.

```
LANE  lane
  X1  Y1  w1   r1   $ start segment 1
  X2  Y2  w2   r2   $ end segment 1, start segment 2
  ....
LANE...
.....
.....
END
```

where:

LANE	Keyword marking start of new lane.
lane	Lane number.
Xn Yn	Global coordinates of point defining start or end of a segment.
w	Width of lane at (Xn,Yn) – required for distributed or full-width line loads, otherwise not significant.
r	Radius of curvature of segment starting at (Xn,Yn). 0 = straight; +ve = curves to left; -ve = curves to right.



LANE DEFINITION

Area Definition

Areas are defined by a list of points specified in order around the boundary. There is one AREA block for each area, followed by an END statement.

```
AREA na
  X1  Y1
  X2  Y2
  .....
AREA...
.....
.....
END
```

where:

AREA	Keyword marking start of new area.
na	Area number.
Xn Yn	Global coordinates of point on boundary of area.

Load Cases – Grillages

The load section of the HBL file consists of one or more CASE blocks followed by an END statement.

```
CASE  n  title
  [OUTPUT]
  GMAX gmax
  [Load type statements]
  [OFFSET statements]
CASE...
.....
.....
END
```

where:

CASE	Keyword marking start of new load case.
n	Load case number.
title	Load case title, up to 50 characters.
OUTPUT	Keyword that specifies output of this load case to the Microstran job. This allows a logical arrangement of the load data in the HBL file with full control over the number of load cases to be analysed. For example, if an assessment has to be made of the effects of trucks in different spans of the bridge, each truck could be assigned to a separate load case but output could be restricted to those combinations with individual cases having a particular offset.
gmax	A dimension that controls the modelling of distributed loads as sets of point loads. A line load is modelled as a set of point loads, each of which is at the midpoint of a short line segment whose length is equal to or slightly less than gmax. An AREA load is modelled as a set of point loads, each of which is

	centred on a rectangle with sides equal to or slightly less than gmax. The recommended value of gmax is about 25% of the minimum grid spacing.

Once defined, the value of gmax is retained for subsequent load cases until the next GMAX statement.

The load type and OFFSET statements are described below.

Load Types – Grillages

Overall Load – Grillages:

```
WUDL  load
```

where:

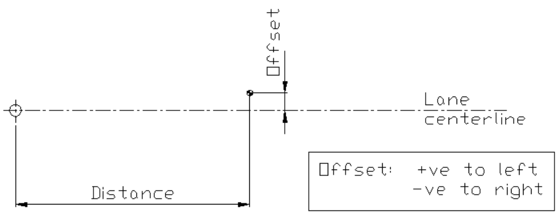
load	Load per unit area over whole grillage.

Point Load – Grillages:

```
POINT  lane  distance  offset  load
```

where:

lane	Lane number.
distance	Distance along the lane centre-line from the lane origin to the load..
offset	Lateral distance of load from lane centre-line – a positive value indicates that the offset is to the left of the centre-line when looking in the direction of increasing distance.
load	The load value.



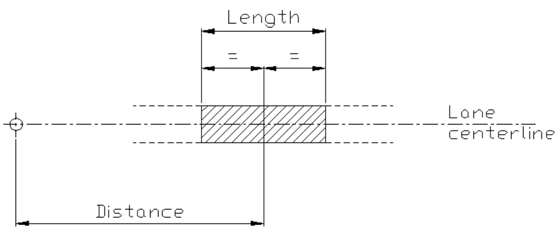
POINT LOAD

Full-Width Lane Load – Grillages

```
LANEL  lane  distance  length  load
```

where:

lane	Lane number.
distance	Distance along the lane centre-line from the lane origin to the centre of the load.
length	Length of load.
load	Load per unit area.



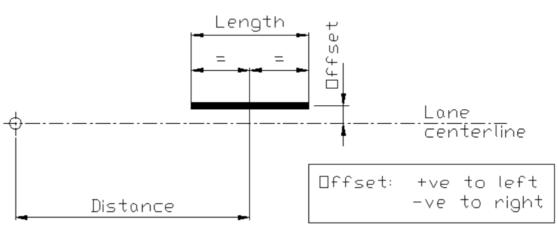
FULL-WIDTH LANE LOAD

Longitudinal Line Load – Grillages

```
LINEL  lane  distance  offset  length  load
```

where:

lane	Lane number.
distance	Distance along the lane centre-line from the lane origin to the centre of the load.
offset	Lateral distance of load from centre-line – a positive value indicates that the offset is to the left of the centre-line when looking in the direction of increasing distance.
length	Length of load.
load	Load per unit length.



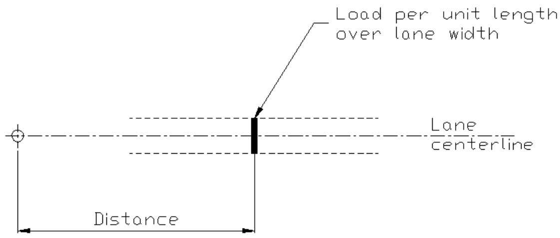
LONGITUDINAL LINE LOAD

Full-Width Lateral Line Load – Grillages:

```
KNIFE  lane  distance  load
```

where:

lane	Lane number.
distance	Distance along the lane centre-line from the lane origin to the centre of the load.
load	Load per unit length.



FULL-WIDTH LATERAL LINE LOAD

Area Load – Grillages

```
AREAL  area  load
```

where:

area	Area number.
load	Load per unit area.

User-Defined Trucks – Grillages

User-defined trucks or other point load groups may be defined as a table of coordinates and load values. Each user-defined truck has its own x-y coordinate system, used to locate its wheels. The truck origin is the point that is the origin of the truck coordinate system. The truck is positioned along a lane with its positive x axis parallel to the lane centre-line and in the direction of increasing distance along the lane.

```
TRUCK name
x  y  P   .....
.....
END
```

where:

name	Character string of one to eight characters used to identify the load group.
x y P	Coordinates and load value. A number of (x y P) groups may be entered on each line.

User-Defined Truck Example

The diagram below shows each wheel of a user-defined truck positioned on the centre-line of a lane 3.7 m wide with the front axle adjacent to the global origin. Each wheel load is 24 kN.

USER-DEFINED TRUCK IN LANE

The .hbl file below contains the coordinates of each wheel with respect to the centre of the front axle. This establishes the truck origin at that point. Any point could be used but using a point on the centre-line of the truck means that the lateral offset dimension on the GROUP statement is zero when the truck is on the lane centre-line.

The OFFSET statement creates 28 new cases, each with the truck moved 0.5 m along the bridge.

```
LANE 1
 -5 -1.85 3.7 0
 15 -1.85 3.7 0
END

TRUCK MyTruck
  0.000  0.900 -24 $ front left
  0.000 -0.900 -24 $ front right
 -3.700  1.000 -24 $ middle left outer
 -3.700  0.800 -24 $ middle left inner
 -3.700 -0.800 -24 $ middle right inner
 -3.700 -1.000 -24 $ middle right outer
 -4.900  1.000 -24 $ rear left outer
 -4.900  0.800 -24 $ rear left inner
 -4.900 -0.800 -24 $ rear right inner
 -4.900 -1.000 -24 $ rear right outer
END

CASE 1 MyTruck - Base case
 OUTPUT
 GMAX 0.25
 GROUP 1 5.0 0.0 MyTruck 0 1

CASE 2 MyTruck
 OUTPUT
 OFFSET 1 0.5 0.5 28 1

END
```

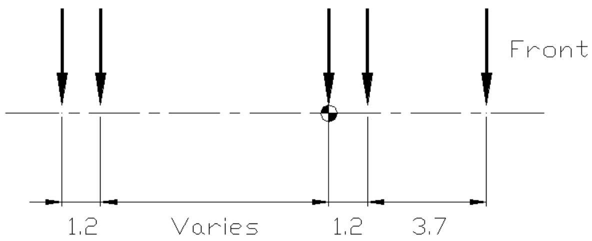
HBL FILE

Truck Loading – Grillages:

```
GROUP lane  distance  offset  truck  var_space  factor
```

where:

lane	Lane number.
distance	Distance along the lane centre-line from the lane origin to the origin of the group or truck load.
offset	Lateral distance of truck origin from centre-line – a positive value indicates that the offset is to the left of the centre-line when looking in the direction of increasing distance.
truck	Truck name – either standard truck name or user-defined truck name.
var_space	Standard truck loading parameter. Ignored if not applicable (e.g. user-defined truck).
factor	Factor applied to truck loads (e.g. impact factor).



T44F TRUCK LOADING

Offset Load Cases – Grillages

New load cases may be generated by offsetting previously defined load cases with the OFFSET statement. Any load case containing offset cases must not itself be offset. There may be more than one OFFSET statement in each CASE block. Each generates at least one load case, whose number is the number on the preceding CASE statement. A series of additional load cases is generated according to the parameters, noff and lc_inc. These parameters are the same for all OFFSET statements in any CASE block and they may be omitted for all but the first OFFSET statement in each CASE block.

```
OFFSET  lcase  start  inc [ noff  lc_inc ]  [F  f]
```

where:

lcase	Number of previously defined load case to be offset.
start	Starting distance along centre-line for loads in lcase.
inc	Incremental distance along centre-line for each additional generated load case. May be zero if noff is zero or if all generated loads are at the same location.
noff	Number of offset cases to be generated, apart from the first. Zero if only one new load case is required.
lc_inc	Load case number increment for additional generated load cases. May be zero if noff is zero.
F	Keyword to specify factor for the offset load case.
f	Factor applied to generated loads.

No other load type may be included in a CASE block containing an OFFSET statement. WUDL and AREAL loads are fixed and will not be offset if included in a load case referred to by an OFFSET statement.

Distribution of Loads to Grillage

Point loads, area loads, and line loads are modelled with statically equivalent point loads applied to the nodes of adjacent members. Overall UDLs are applied as a series of linearly varying distributed member loads. For areas enclosed by three or four members, the load is allocated by bisecting the angles between adjacent members. For a rectangular grid this gives a set of triangular and trapezoidal loads along the short and long members respectively. For areas with more than four distinct vertices, uniform loads are applied to each bounding member.

No loads will be allocated to any cantilevers, and the grillage must not contain members that cross without intersecting.

Lane loads that have a lateral offset value (POINT, LINEL, and GROUP loads) may be applied anywhere across the grillage, not necessarily within the defined lane width. Where these are the only loads applied,

and all potential lanes are parallel, only a single lane definition is required (e.g. the bridge centre-line).

Example – HBL File

```
Curved Bridge Deck

LANE  1  $ Deck centre-line
$    X          Y      Width Radius
  537.133    461.400    3.0    0     $ start of lane s=0
  531.793    485.830    3.0    0     $ start of deck s=25m
  531.259    488.273    3.0    0     $
  530.704    490.710    3.0    0     $
  530.129    493.143    3.0    0     $
  529.534    495.571    3.0    0     $
  529.061    497.440    3.0    0     $
  528.260    500.406    3.0    0     $
  527.552    502.804    3.0    120   $ 120m radius to next point
  479.171    567.624    3.0    0     $
  474.948    570.301    3.0    0     $
  470.618    572.800    3.0    0     $
  466.187    575.116    3.0    0     $
  461.664    577.245    3.0    0     $
  457.056    579.185    3.0    0     $ end of deck s=150m
  433.916    588.645    3.0    0     $ end of lane s=175m
END

CASE 1  T44F s=25 (start of deck)
 $OUTPUT
 GMAX 0.5  $ spacing of output point loads
 GROUP 1  25  1.875 T44F 3.0 1.0 $ lane distance offset truck var_space factor

CASE 2  T44B s=105R
 $OUTPUT
 GMAX 0.5  $ spacing of output point loads
 GROUP 1 105 -1.875 T44B 3.0 1.0 $ lane distance offset truck var_space factor

CASE 3  KNIFE LOAD s=25 (start of deck)
 $OUTPUT
 KNIFE 1  25  -50               $ lane  distance  load

CASE 4  HB40 s=130
 $OUTPUT
 GMAX 0.5  $ spacing of output point loads
 GROUP 1 130 -1.875 HB40 6.0 1.0 $ lane distance offset truck var_space factor

CASE 5  T44F s=60
 $OUTPUT
 GMAX 0.5  $ spacing of output point loads
 GROUP 1  60  1.875 T44F 3.0 1.0 $ lane distance offset truck var_space factor

CASE 6  Two trucks  40L 105R
 OUTPUT
 OFFSET 1  15 0 0 0  $ lcase start inc noff lc_inc
 OFFSET 2   0 0      $ lcase start inc

CASE 7  Two trucks  60L  85R
 OUTPUT
 OFFSET 1  35 0 0 0  $ lcase start inc noff lc_inc
 OFFSET 2 -20 0      $ lcase start inc

$CASE 8  Two trucks  80L  65R
$ OUTPUT
$ OFFSET 1  55 0 0 0 $ lcase start inc noff lc_inc
$ OFFSET 2 -40 0     $ lcase start inc

END
```

On installation of Microstran, this example is included in the examples folder as Deck1. The diagram on page 199 shows the structure and the generated loads.

Moving Loads Graphics

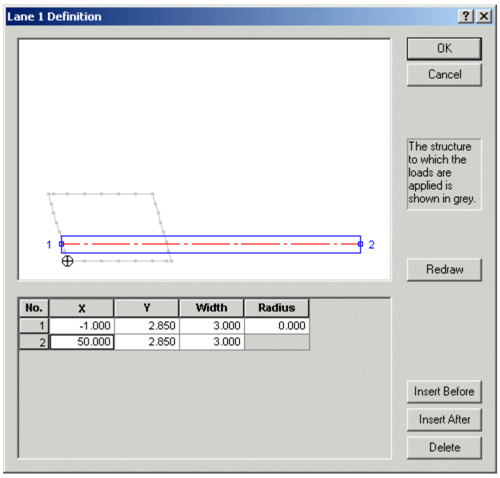
The graphical interface simplifies the input of data for moving loads on a grillage. To generate the required loads, select the **Loads > Moving Loads > Grillage** command, then select the load-carrying members of the grillage.

Note: Making a set of the grillage members to be loaded allows them all to be selected from the Current Set pull-down box in the View toolbar.

The steps required to run a simple example are shown below. A small skew grillage has been defined with an existing load case and load combinations are to be generated.

Step 1 – Specify lanes

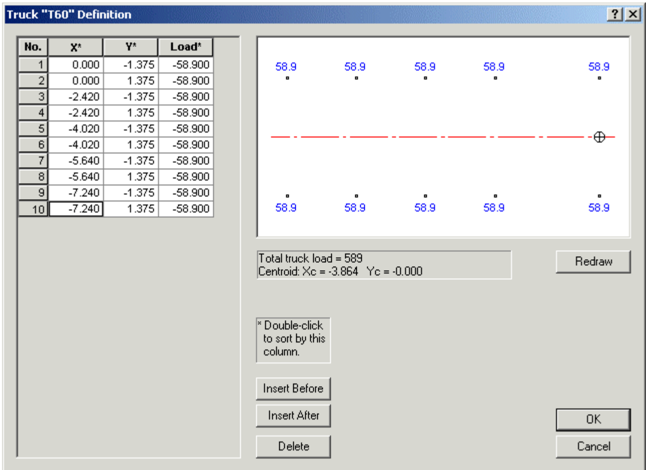
Use the **Loads > New Lane** or **Loads > Edit Lane** command to specify the lanes, allowing sufficient runoff for trucks.



The **Loads > Display Lanes** command can now be used to see the lane centre-lines superimposed on the grillage. Press the space bar to show the next lane.

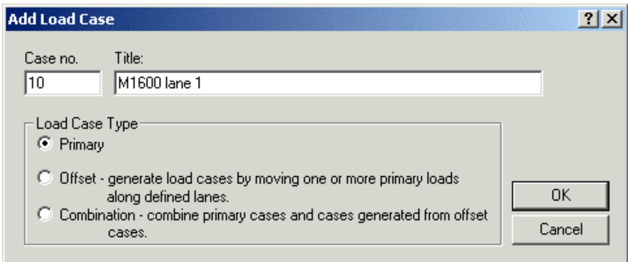
Step 2 – Specify user-defined truck

Use the **Loads > New Truck** or **Loads > Edit Truck** command to specify user-defined trucks as required. The dialog box below shows the specification of a truck named T60.

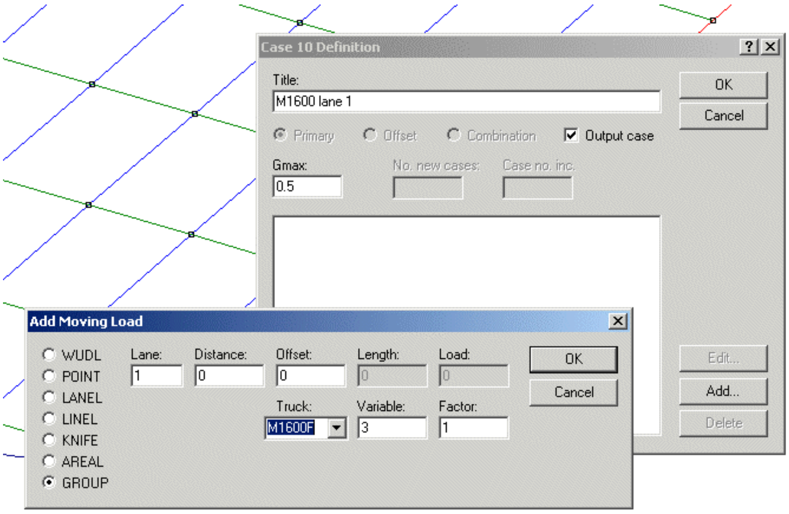


Step 3 – Set up M1600 trucks at the start of lanes 1 and 2

Use the **Loads > New Load Case** command to specify primary load cases 10 and 11 with standard M1600 trucks at the start of lanes 1 and 2, respectively.

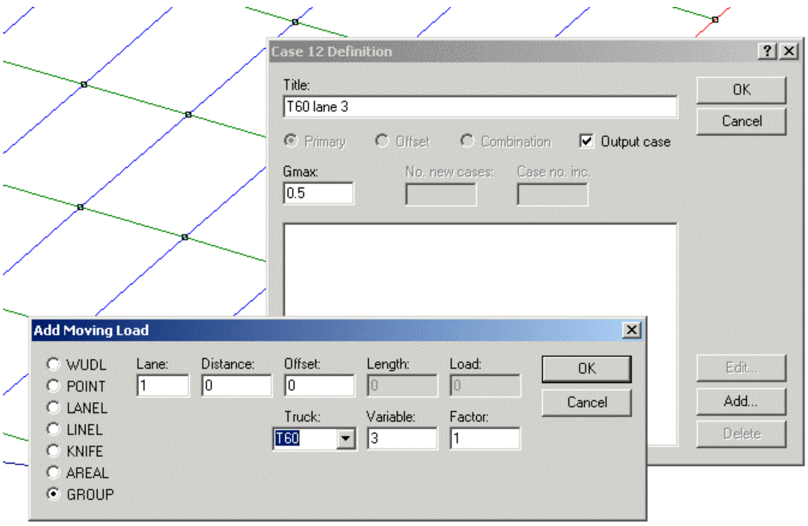


In the case definition dialog box click the **Add** button to specify loads. Then, in the Add Moving Load dialog box select a GROUP load, specifying an M1600F truck (i.e. a forward-facing M1600 truck) at the start of lane 1.



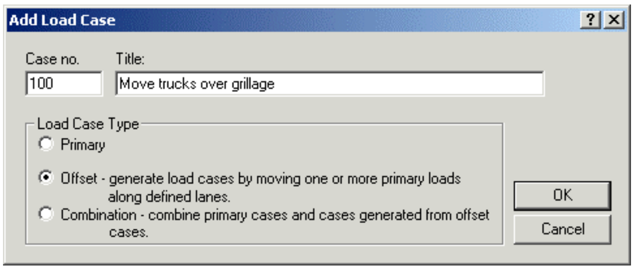
Step 4 – Set up user-defined truck at start of lane 3

Define primary load case 12, the user-defined truck at the start of lane 3.

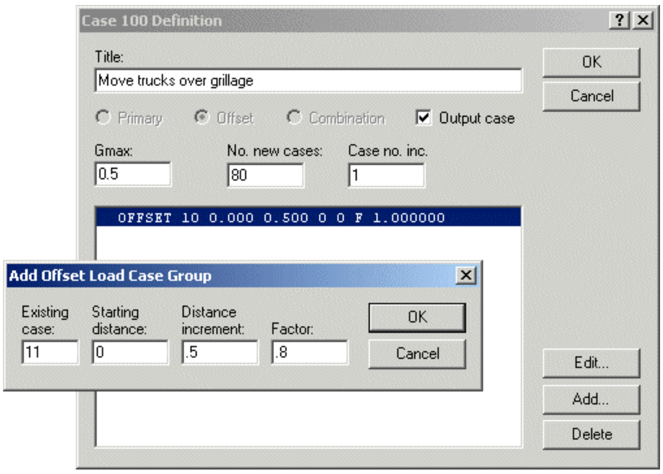


Step 5 – Specify offset load case to move trucks across grillage

Use the **Loads > New Load Case** command to define offset load case 100 to move all the trucks across the grillage.



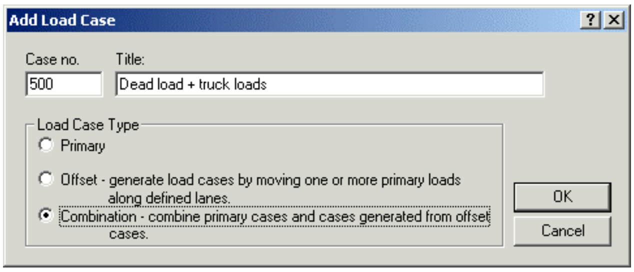
Click the **Add** button in the case definition dialog box to specify the cases that are going to be offset. For each case the starting distance and the offset data are specified. In the image below the offset data for case 10 has been input and it is shown in the list box. The offset data for case 11 is specified in the small dialog box and it will appear in the list box when the **OK** button is clicked. Offset data for case 13 is input similarly.



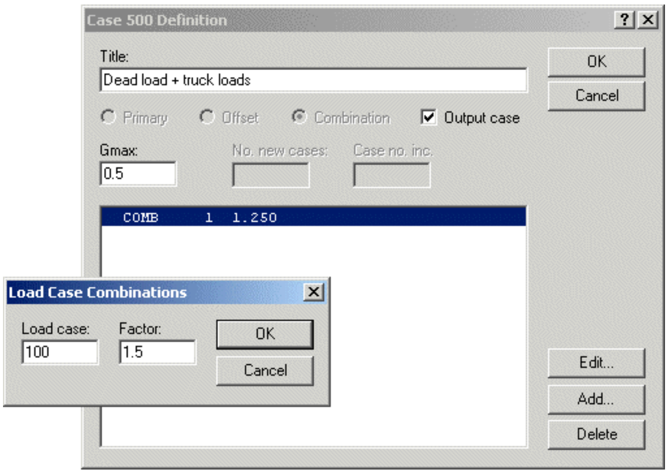
Primary load cases 100 to 180 will be generated.

Step 6 – Specify combination load cases

Use the **Loads > New Load Case** command to define combination load case 500 as a combination of case 1 with a factor of 1.25 and offset case 100 with a factor of 1.5.



Click the **Add** button in the case definition dialog box to specify the combinations. In the image below the combination data for case 1 has been input and it is shown in the list box. The combination data for case 100 is specified in the small dialog box and it will appear in the list box when the **OK** button is clicked. Combination load cases 500 to 580 will be generated using case 1 and offset cases 100 to 180.

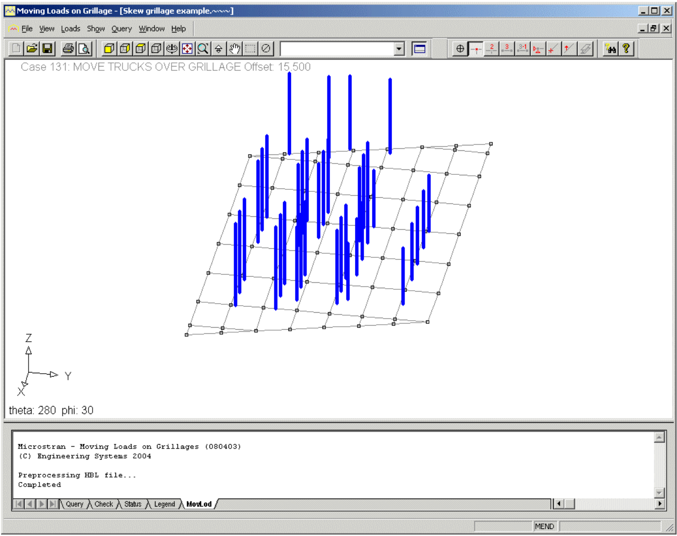


Step 7 – Process applied loads

The data input in the foregoing steps is automatically stored as instructions in the HBL file. On selecting the **Loads > Process Applied Loads** command these instructions are executed, generating new load cases in the Microstran database.

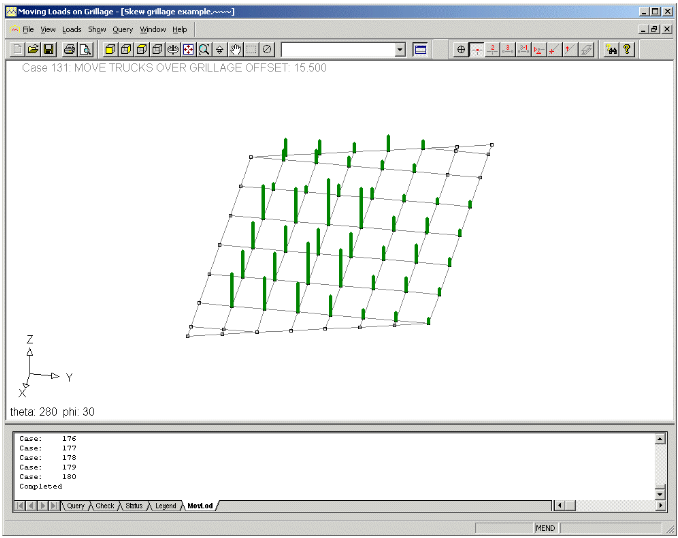
Step 8 – Display applied loads

Use the **Loads > Display Applied Loads** command to show the loads one case at a time. The first load case is shown; press the space bar to advance to the next case. Select the **Loads > Drive** command to show all cases cyclically. This gives the appearance of the loads moving, i.e. the trucks advancing along their lanes. Select the **Drive** command again to stop the moving loads display.



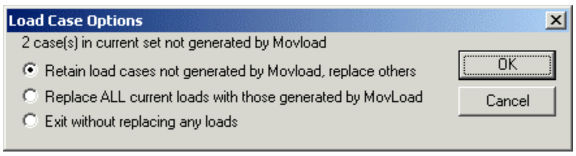
Step 9 – Distribute loads to grillage

The **Loads > Distribute Loads to Grillage** command calculates and applies loads to the nodes in the grillage. As in the previous step, you may display the loads on the grillage by using the **Loads > Display Applied Loads** command.

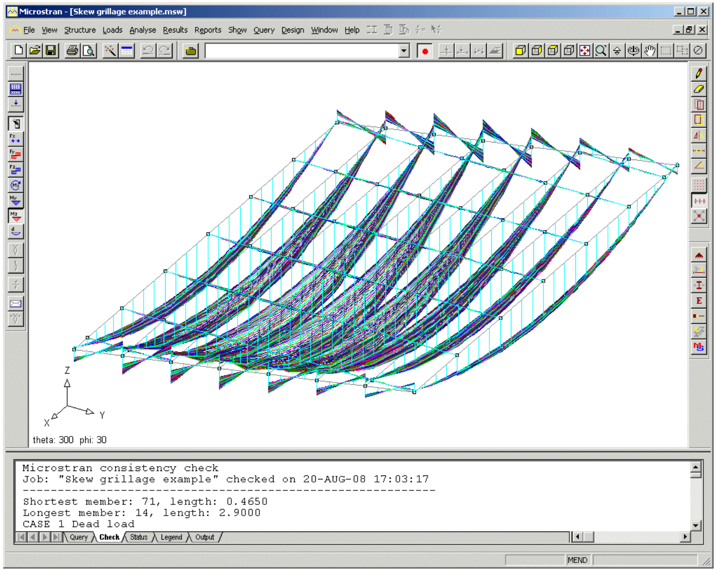


Step 10 – Update Microstran Loads

Close the Movload window by selecting the **File > Exit** command or clicking the **X** button at the top right corner. The Microstran window should now be visible with the Load Case Options dialog box displayed. Original load cases that are not included in the HBL file may be retained.



The Microstran database should now include the loads generated by the Movload module and the updated model may be analysed. The image below shows the bending moment diagram for all the combination load cases.



12:Input Tutorials

This chapter contains several tutorial examples to show you how to go through all the steps needed to do a typical job. Refer to Chapter 2 – "Getting Started" for an explanation of Microstran basics.

Some of the tutorial examples are available from the Microstran website as self-playing demo files. You can download these and view them on your computer if you wish.

Tutorial 1 – Running an Existing Job

In this tutorial, the portal frame example included on the distribution diskettes is run to demonstrate analysis, screen plots, and how loads can be changed graphically. After installing Microstran successfully, the input data for this job will exist in your data folder, in the form of an archive file. Input data, results and diagrams for this example are included in Chapter 20 (see "Example 2 – Portal Frame" on page 345).

Step 1 – S*tart Microstran.*

Click on the Windows **Start** button, select **Programs** and then **Microstran**. Maximize the Microstran window if it is too small.

Step 2 – *Import a file.*

Select **File** on the menu bar; select **Import**; select **Archive File**; select Example 2.arc, from the listed files and click on the **Open** button. You should now see the portal frame displayed in the Microstran window. Click the **Display Node Numbers** and **Display Member Numbers** buttons on the Display toolbar at the top of the window and notice the display change.

Step 3 – *Analyse the structure.*

Select **Analyse** on the menu bar; select **Linear**. Notice a new window open as the analysis module performs the analysis. You should see the message "Linear analysis completed" at the bottom of the window. Close the analysis window by clicking on the **X** button at the top right of the window.

Step 4 – ***Display applied loads.***

Click on the **Display Applied Loads** button at the top of the Results toolbar on the left of the main window. In the list box showing the load cases titles select cases 1, 2, 3, and 4 (the primary load cases); click on the **OK** button and notice that the loads are now displayed. Applied loads are not shown for combinations. Click on the **Display Applied Loads** button again to turn off the load display.

Step 5 – ***Display bending moment diagrams.***

Select **Results** on the menu bar; select **Select Load Cases**; deselect the primary load cases and select case 5; click on the **OK** button in the dialog box; click on the **Bending Moment Mz** button. Click on the **Display Results Values** button on the Display menu; notice bending moment values now displayed on the bending moment diagram. If the bending moment diagram does not fit within the window click on the **Zoom Out** button on the View menu.

Step 6 – ***Change scale of plot (optional).***

Select **View** on the menu bar; click on **Display Options**; click on the **Scales** tab of the Display Options property sheet; press the Tab key until the **Bending moments** scale is selected; type **2**; press **Enter**. Notice that the bending moment diagram is redrawn with the new scale factor. To display the values at the extremities of the plot you may have to click on the **Zoom Out** button.

Step 7 – ***Preview a plot and print it.***

Click on the **Print Preview** button on the main toolbar; notice the graphics preview display; click on the **Close** button. Click on the **Print View** button on the main toolbar. Type any desired title for the view in the dialog box; click on **OK**. The view should now print on the default printer, exactly as it was shown in the print preview. It should be very similar to the examples in Chapter 20. Turn off the bending moment display by clicking on the **Bending Moment Mz** button again.

Step 8 – ***Change the value of a load.***

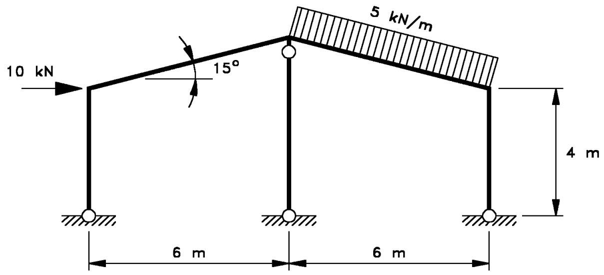
Select **Loads** on the menu bar; select **Select Input Case**; click on case 4; click on the **OK** button. Notice that the loads are displayed for case 4 even though the **Display Applied Loads** button is not depressed. When in load input mode the current input case is displayed automatically. Select **Delete Load** on the **Load** menu; select **Member Loads**; click on member 3, the right rafter; click the right mouse button; click **OK** on the context menu (this confirms selection of member 3); right-click to terminate the command. Notice that the load on member 3 has now disappeared. Select the **Loads > Member Loads** command; click on member 3; confirm the selection after right-clicking; click on the uniform load in the dialog box; type a value of -2.5 in the **F1 =** edit box; check that **Type** is **FY** and that **Axes** are **Local**; click on the **OK** button or press **Enter**; right-click to terminate the command. Notice the new load on member 3. Notice also that the buttons on the Results toolbar

have become greyed out because the analysis results were invalidated when the load data changed.

Step 9 – ***Exit Microstran.***

Click on the **X** button at the top right of the Microstran window. In the dialog box click on the **Yes** button to save the job.

Tutorial 2 – Running a New Job



TUTORIAL 2

This tutorial takes you through the steps needed to input a structure and loads graphically, select sections and material, analyse the structure, and display the results. The structure and loads are shown in the diagram above.

Step 1 – ***Start Microstran.***

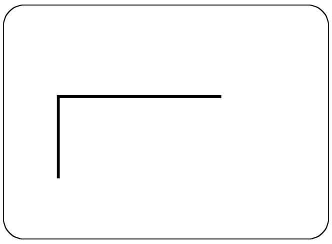
Click on the Windows **Start** button, select **Programs** and then **Microstran**. Maximize the Microstran window if it is too small.

Step 2 – ***Set the job name, titles etc.***

Select **New** on the **File** menu or click on the **New Job** button; click **OK** to accept the default job file folder; type "TUT2" for the job name (do not include the quotes); click on **OK** or press **Enter**. Type a job title and press **Enter**. Check that the **Plane frame** and **Y axis** option buttons are selected; click on **OK**. Select **m, kN, t, °C** for units if not already selected; click on **OK**.

Step 3 – ***Draw left-hand column and a horizontal rafter.***

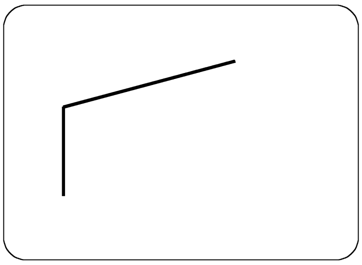
Click on the **Snap to Grid** button on the Draw toolbar at the right of the main window; click on the **Front View** button on the View toolbar at the top of the window. Click on the **Draw Members** button (the one with the pencil); type "0" or "0,0" and then press **Enter**; move the mouse to draw a vertical member 4 m high and then click; move the mouse right to draw a horizontal member 8 m long and then click; click the right mouse button and on the context menu select **End Line**.



TUTORIAL 2 – STEP 3

Step 4 – *Rotate rafter 15°.*

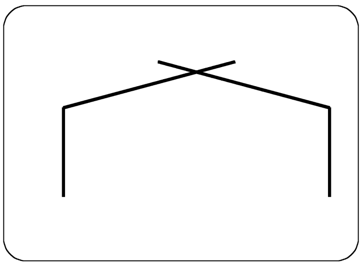
Click on the **Rotate** button; click on the horizontal member; right-click, and on the context menu select **OK**; click on the top left node (the centre of rotation); check that the axis of rotation is **Z**; type "15" for the angle, and press **Enter**. Right-click to terminate the command.



TUTORIAL 2 – STEP 4

Step 5 – *Reflect column and rafter.*

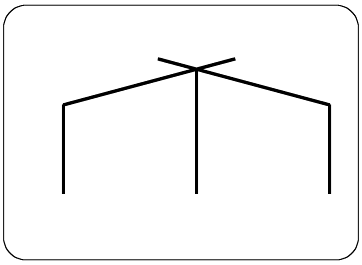
Click on the **Reflect** button; click on both members; right-click and click on **OK**; press **Enter**; click on any grid point 6 m to the right of the column; check that the normal axis is **X** and that the **Extrude nodes** box is not checked; click on **OK** or press **Enter**. Right-click to terminate the command.



TUTORIAL 2 – STEP 5

Step 6 – ***Draw central column.***

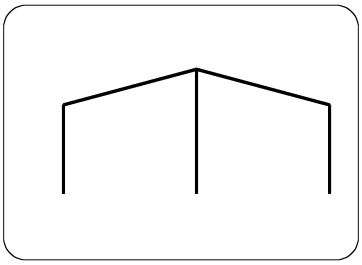
Click on the **Draw Members** button; referring to the coordinate readout in the status line at the bottom right of the window, click on point (6,0); click on the **Snap to Intersection** button on the Draw toolbar at the right of the window; click on the apex node; right-click and select **End Line** on the context menu.



TUTORIAL 2 – STEP 6

Step 7 – ***Trim overhangs on rafters.***

Click on the **Erase Members** button; click on each rafter overhang; right-click and select **OK** on the context menu. Right-click to terminate the command.



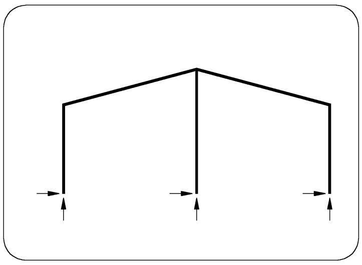
TUTORIAL 2 – STEP 7

Step 8 – ***Display attributes.***

Select the **View > Display Options** command; click on the **Nodes** tab; click in the check boxes for **Node numbers**, **Node symbols**, and **Supports**. Click on the Members tab and check the boxes for **Member numbers**, **Member axes**, and **Pins**. Click on the **OK** button and notice these items now displayed in the view.

Step 9 – ***Input supports.***

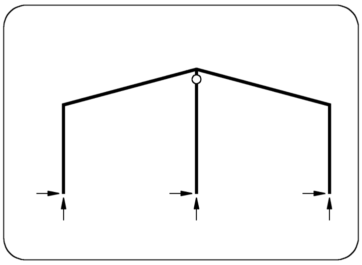
Select the **Structure > Attributes > Fixed Supports** command; click on each of the three support nodes; right-click and then click the **OK** button; click on the pinned support button and then on the **OK** button; right-click to terminate the command; select the **View > Redraw** command. Notice the symbols now displayed at the support nodes. (Instead of using the menu item, you could have clicked on the **Input Supports** button on the Attributes toolbar.)



TUTORIAL 2 – STEP 9

Step 10 – ***Input pin at top of central column.***

Click on the **Input Releases** button on the Attributes toolbar; click on the central column; right-click and select **OK**; click on the button with the horizontal pin axis in the **End "B"** group; right-click to terminate the command; select the **View** > **Redraw** command. Notice the small circle plotted at the top of the central column to represent the pin.



TUTORIAL 2 – STEP 10

Step 11 – ***Input section numbers.***

Click on the **Input Section Number** button on the Attributes toolbar; click on each rafter; right-click and select **OK**; click on the up spin button so the section number becomes "2" and click on **OK**; click on the central column; right-click; click on **OK**; input "3" for the section number and click on **OK**; right-click to terminate the command. Notice the colours of the rafters and the central column have changed to represent the new section numbers. Click on the Legend tab of the output window at the bottom of the main window and notice the key to the colours of the sections.

Step 12 – ***Input sections from library.***

Click on the **Input Section Properties** button on the Attributes toolbar; click on the **OK** button in the Section dialog box. Notice the columns (section 1) highlighted. Select the **Library** option button and click on **Input/Edit**; double-click on **UB**; select a 200 mm UB section; click on **OK**. Notice that the rafters are now highlighted (section 2). Click on the **OK** button in the Section dialog box. Select a 300 mm UB section. Using the above method input a 200 mm UB for the central column.

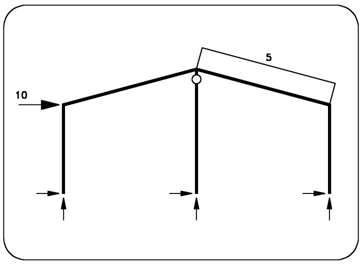
Step 13 – ***Input material from library.***

Select the **Structure > Attributes > Material Properties** command. Notice the whole structure is highlighted indicating that all members have a material number of “1”. Click on the **OK** button in the Material dialog box. Check that the **Library** option button is selected; click on **OK**; double-click on **STEEL**.

Step 14 – ***Input loads.***

Select the **Loads > Add Case** command; type “Wind load”; click on **Accept**; click on **OK**; select the **Loads > Node Loads** command; click on the top left node; right-click and select **OK** on the context menu; type “10” in the **X** edit box; press **Enter**; right-click to terminate the command. Notice the arrow indicating a node load. Click on the **Display Load Values** button on the Display toolbar and notice the arrow is now labelled with the value of the load.

Select the **Loads > Member Loads** command; click on the right-hand rafter; right-click and select **OK** on the context menu; click on the **Uniform** button; type “-5” in the **F1 =** edit box; check that the **Type** combo box is set to **FY** and that the **Local** option button is selected.



TUTORIAL 2 – STEP 14

Step 15 – ***Analyse.***

Select the **Analyse > Linear** command. Close the analysis program window by clicking on the **X** button when the analysis is completed.

Step 16 – ***Display results.***

Click on the **Bending Moment Mz** button on the Results toolbar. Click the **All** button and the **OK** button in the load case selection dialog box. The bending moment diagram should now be displayed. Click the **Display Results Values** button on the Display toolbar and the bending moment diagram will be labelled with bending moment values.

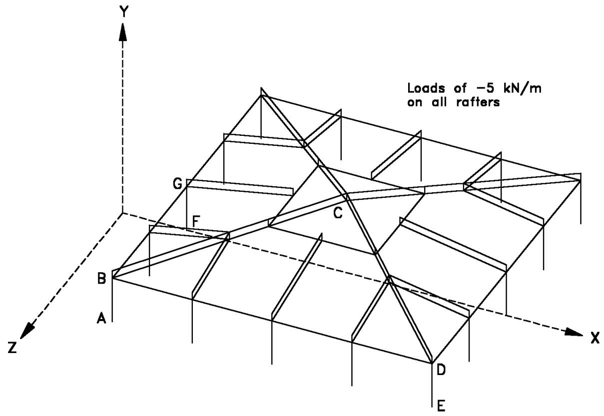
Step 17 – ***Use query function to display results numerically.***

Select the **Query > Member Forces** command. Click on the right rafter and notice the results in the **Query** page of the output window at the bottom of the main window. This may be scrolled if the results you want are not shown. *Press the space bar and notice how the highlight moves to the next member in the numbering sequence.* The results are now displayed for this member.

Step 18 – *Save Job and exit Microstran.*

Click on the **Save Job** button (the one with the diskette); click on the **Save** button in the dialog box; select the **File > Exit** command.

Tutorial 3 – A 3-D Example



TUTORIAL 3 – A 3-D EXAMPLE

In this tutorial we input the structure and loads shown in the above diagram. There are many different ways of creating a model such as this and this tutorial illustrates a few interesting techniques.

With reference to the diagram, the method adopted is to input a quarter of the structure, ABCDE, and then to perform a polar copy about the vertical axis. The side dimension is 20 m, the eaves height is 3 m, and the apex height is 5 m. Initially, the column, AB, and the rafter, BC, are drawn in the XY plane, the column is then moved 10 m in the +Z direction, and then the rafter and column are reflected about a plane parallel to the YZ plane.

Step 1 – ***Start Microstran.***

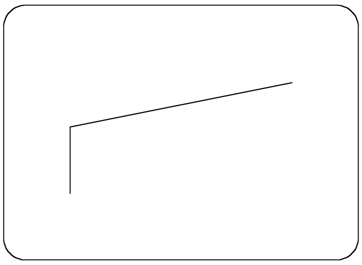
Click on the Windows **Start** button, select **Programs** and then **Microstran**. Maximize the Microstran window if it is too small.

Step 2 – ***Set the job name, titles etc.***

Select **New** on the **File** menu or click on the **New Job** button; click **OK** to accept the default job file folder; type "TUT3" for the job name (do not include the quotes); click on **OK** or press **Enter**. Type a job title and press **Enter**. Check that the **Space frame** and **Y axis** option buttons are selected; click on **OK**. Select **m, kN, t, °C** for units if not already selected; click on **OK**.

Step 3 – ***Draw column and rafter in XY plane.***

Click on the **Snap to Grid** button on the Draw toolbar at the right of the main window; click on the **Front View** button on the View toolbar at the top of the window. Click on the **Draw Members** button; click the mouse on any grid point at the lower left of the screen; move the mouse to draw a vertical member 3 m high and then click; type "R10,2". Notice that the Node Coordinates dialog box has appeared and now displays your keystrokes. Press **Enter** and notice that a rafter has been drawn to the point 10 m to the right and 2 m up from the top of the column. Click the right mouse button and on the context menu select **End Line**.



TUTORIAL 3 – STEP 3

Note: In this example, the nodes, A, B, and C, fall conveniently on grid points (using the **Structure > Drawing Settings > Grid Spacing** command, the grid spacing may be set in each direction to values other than 1 m). You may draw a member to a point which is not on the grid either by entering the coordinates or relative coordinates of the point instead of attempting to click on it, or by clicking on a nearby grid point and subsequently moving the node so created.

Step 4 – ***Move column to corner, A.***

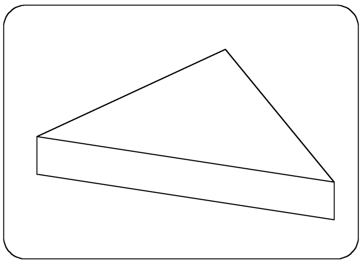
Click the **Move Members** button on the Draw toolbar; click on the vertical member; right-click and select **OK** on the context menu; type "0 0 10" and press **Enter**; click the right mouse button to terminate the command.

Step 5 – ***Reflect column and rafter.***

Click on the **Reflect** button on the Draw toolbar; click on both members; right-click and select **OK** on the context menu. *Notice now that the prompt area of the status bar displays a message "Click on point or enter offset".* Click on the right end of the rafter; in the Reflect dialog box, check that **X** is selected for the normal axis, select **Extrude nodes**, and click on the **OK** button. Right-click to terminate the command.

Step 6 – ***Change viewpoint.***

Click on the **Oblique View** button on the View toolbar. The view has now changed to the default oblique view for a 3-D structure.



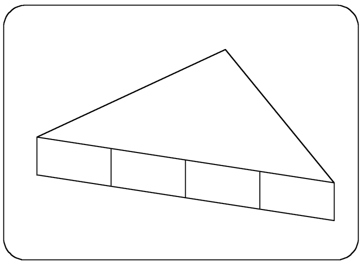
TUTORIAL 3 – STEP 6

Step 7 – ***Select middle/end snap mode.***

Click on the **Snap to Mid/End** button on the Draw toolbar. Notice that the grid now disappears.

Step 8 – ***Draw interior columns.***

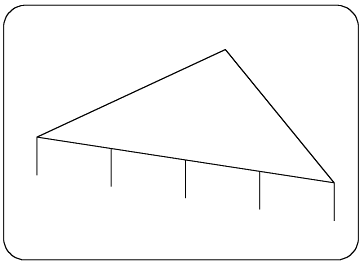
Click on the **Display Node Symbols** button on the Display menu. Notice the appearance of small square symbols at each node. Click on the **Draw Members** button; position the cross-hair cursor near the middle of the lower horizontal member and click; notice that the beginning of a new member snaps to the mid-point of the member; click on the mid-point of the upper horizontal member. Right-click and select **Break Line** on the context menu. The horizontal members have been sub-divided automatically by drawing the column at the mid-points. Now, draw columns at the quarter-points by repeating this procedure. Right-click and select **End Line** on the context menu.



TUTORIAL 3 – STEP 8

Step 9 – ***Erase baseline members.***

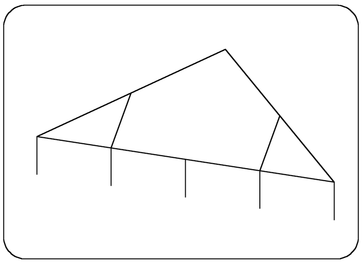
Click on the **Erase Members** button on the Draw toolbar and click on each of the four baseline members. If you accidentally select the wrong member, de-select it by clicking on it again. Right-click and select **OK** on the context menu; right-click to terminate the command. *Note that you can also select members by enclosing them in a selection box created by clicking on a corner and dragging to the diagonally opposite corner.*



TUTORIAL 3 – STEP 9

Step 10 – *Draw outer rafters.*

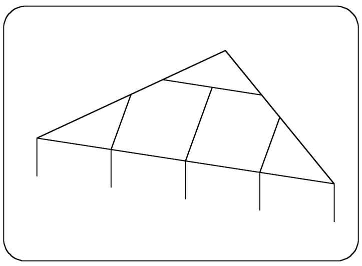
Click on the **Draw Members** button; click on the top of the left interior column and then the mid-point of the left rafter; right-click and select **Break Line**. Repeat for the right rafter but select **End Line**.



TUTORIAL 3 – STEP 10

Step 11 – *Sub-divide upper rafters, draw beam and central rafter.*

Click on the **Sub-divide Members** button on the Draw toolbar; click on the top parts of the left and right rafters; right-click and select **OK**; click on the up spin button so that the number of segments is “3”; click on the **OK** button; right-click to terminate the command. Click on the **Draw Members** button; click on the lower third-point of the left upper rafter; click on the lower third-point of the right upper rafter; right-click and select **Break Line**. Click on the top of the central column; click on the mid-point of the upper beam; right-click and select **End Line**.



TUTORIAL 3 – STEP 11

Step 12 – *Eliminate extra nodes in upper rafters.*

Select the **Structure > Move > Node**; click on the top third-point node on the left upper rafter; click on the apex node; right-click. Repeat this operation for the extra node on the right upper rafter. Select the **View > Redraw** command. Notice that the extra nodes have now gone. (In each case, moving the extra node to one end has stretched one part of the member to the full length and made the other part a zero-length member, which is automatically eliminated.)

Note: The upper part of each rafter was sub-divided into thirds as a convenient means of locating the node at which the upper beam is connected to the rafter. It is not essential that the upper third-point nodes are eliminated. Instead of sub-dividing, a single node could have been introduced into each upper rafter by use of the **Structure > Insert Node** command.

Step 13 – *Input supports.*

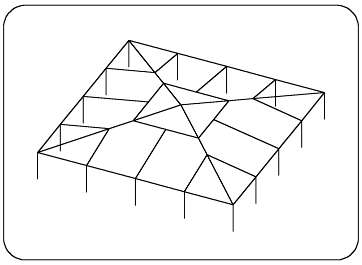
Click on the **Input Supports** button on the Attributes toolbar; click on each of the five support nodes; right-click and select **OK**; click on the pinned support button and then the **OK** button; right-click; click on the **Display Supports** button on the Display menu. Notice the symbols at each support.

Step 14 – *Input section numbers.*

Click on the **Member Numbers** button to turn on member numbering and the **Display Supports** button to turn off support display. Click on the **Input Section Number** button on the Attributes toolbar; click on all parts of the left and right diagonal rafters; right-click and select **OK**; click on the upper spin button and select “2”; click on the **OK** button. Click on the other rafters and beams; right-click and select **OK**; select “3” for the section number and click on **OK**; right-click to terminate the command. Notice the new section numbers on the rafters and beams, and the new colours.

Step 15 – *Perform polar copy.*

Select the **Structure > Copy > Polar** command. Click at the bottom right of the structure and drag to the top left, enclosing the whole structure with the selection box; right-click and select **OK**. Notice the prompt to “Click on centre of rotation or enter coordinates”. Click on the apex node. In the Polar Copy dialog box, select **Y** for **Axis of rotation**, type “90” for **Angle** and “3” for **Number of copies**. Press **Enter**. Right-click to terminate the command. Click on the **Zoom Extents/Limits** button on the View menu to redraw the structure filling the display.



TUTORIAL 3 – STEP 15

Step 16 – *Set orientation of columns*.

Select the **View > Display Options** command; click the **Members** tab; uncheck the **Member numbers** box; check the **Member axes** box; click on **OK**.

Note: The *y* axes of all columns point to the left (-X) while the *y* axes of all roof members point up (+Y). These are default orientations. We require that corner columns are rotated 45° and that other front and rear columns be rotated 90°. When setting the orientation of the corner columns, the apex node will be used as a "reference node" (see "Member Definition" on page 83).

Click the **Input Reference Node/Axis** button on the Attributes toolbar; click on each of the corner columns; right-click and select **OK**. Notice the prompt "Select reference node or enter node/axis". Click on the apex node; click on **OK** in the Reference Node/Axis dialog box. Click on each of the other front and rear columns; right-click and select **OK**; type "Z" and press **Enter**; right-click to terminate the command. Click on the **Zoom Extents/Limits** button to redraw the view. Notice that the columns are now oriented as required.

Select the **View > Display Options** command; click the **Members** tab; uncheck **Member axes** box; click on **OK**.

Note: There are some advantages in steel design in having the simple convention for column orientation that all column *y* axes point outwards. To achieve this, you could readily use both X and -X and Z and -Z as reference axes for columns not on the corners.

Step 17 – *Input sections from library.*

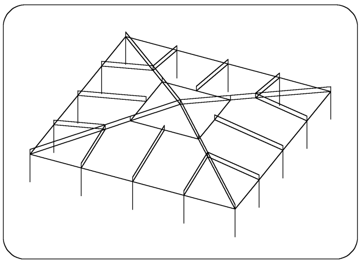
Click on the **Input Section Properties** button; click on **OK** for section 1; click on **OK**; click on **UB**; select a 250 mm UB section; click on OK. Section 1 has now been input. Repeat the process, selecting a 200 mm UB for section 2 and a 300 mm UB for section 3. Click on the **Render** button on the Display toolbar and notice the section shapes have been displayed. Click the button again to turn off rendering.

Step 18 – *Input material from library.*

Select the **Structure > Attributes > Material Properties** command. Notice the whole structure is highlighted indicating that all members have a material number of "1". Click on the **OK** button in the Material dialog box. Check that the **Library** option button is selected; click on **OK**; double-click on **STEEL**.

Step 19 – *Input loads.*

Select the **Loads > Add Case** command; type “Design case”; click on **Accept**; click on **OK**; click on the **Front View** button. Select the **Loads > Member Loads** command; click at the left of the view and drag to the top right, enclosing all roof members; select another box that encloses just the eaves beams; select another box enclosing just the upper beams. Notice that all rafters have now been selected. Right-click and select **OK**; click on the **Uniform** button; type “-5” in the **F1 =** edit box; check that the **Type** combo box is set to **FY** and that the **Global** option button is selected. Right-click to terminate the command. Click on the **Oblique View** button.



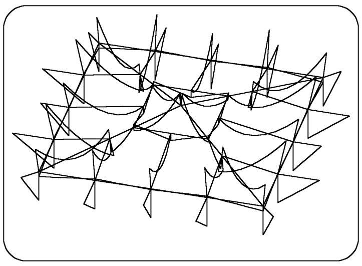
TUTORIAL 3 – STEP 19

Step 20 – *Analyse.*

Select the **Analyse > Linear** command. Close the analysis program window by clicking on the **X** button when the analysis is completed.

Step 21 – *Display results.*

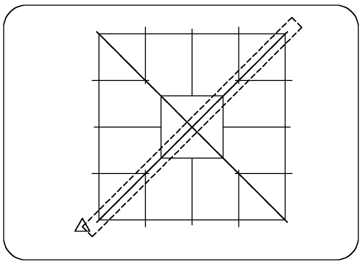
Click on the **Bending Moment Mz** button on the Results toolbar. Click the **All** button and the **OK** button in the load case selection dialog box. The bending moment diagram should now be displayed. Click the **Display Results Values** button on the Display toolbar and the bending moment diagram will be labelled with bending moment values.



TUTORIAL 3 – STEP 21

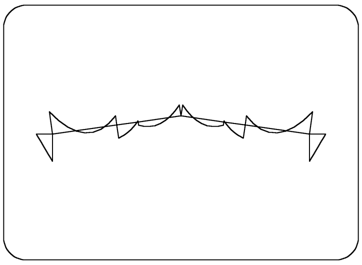
Step 22 – *Display results for diagonal rafter.*

Click on the **Plan View** button; select the **View > Limit > Boundary** command; starting at the bottom left, click the corners of a narrow rectangle enclosing the diagonal rafter. The last point should be clicked near the first point to close the boundary.



TUTORIAL 3 – STEP 22a

Click on the **Front View** button and a few times on the **Viewpoint Left** button to rotate the axes so that the X and Z axes are coincident. Notice that the bending moment diagram displayed is now a view at right angles to the diagonal rafter.



TUTORIAL 3 – STEP 22b

Note: The discontinuities in the bending moment diagram at the points where the beams meet the diagonal rafters. These are caused by bending moments transferred from the beams. The connections between beams and rafters would usually be such that this moment transfer does not occur. You may insert moment releases in the beams to more accurately model this situation.

Step 23 – *Save Job and exit Microstran.*

Click on the **Save Job** button (the one with the diskette); click on the **Save** button in the dialog box; select the **File > Exit** command.

13:CAD Interface

General

The CAD Interface is an integral part of Microstran offering the capability of exchanging 3-D data between a structural model and CAD systems. These functions are selected with the **File > Import > CAD DXF** command and the **File > Export > CAD DXF** command. Structure information is exchanged by means of an AutoCAD DXF.

You can easily create a Microstran model by importing a suitable CAD DXF. In some situations, an architect's drawing files may be used for this purpose. In other situations, it may be expedient to have a drawing prepared in CAD so you can import the structure directly into Microstran. Microstran is compatible with the AutoCAD Release 12 DXF format.

Conversely, when you have a Microstran model, it's possible to export a DXF that could become the basis for a CAD drawing.

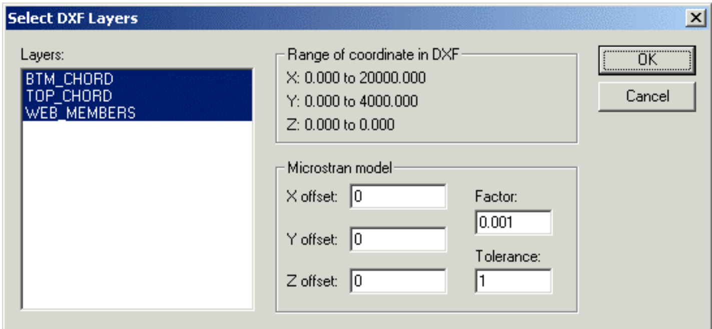
Note: You can use the Windows **Paste** command to transfer any part of a Microstran image into CAD.

Microstran's RC Detailing option creates a DXF to transfer RC details to CAD (for more information, see Chapter 19 – "RC Design & Detailing".

Importing a CAD DXF

Members must be represented on the drawing as straight lines (using the AutoCAD "LINE" entity). Nodes are not represented separately in the DXF but are obtained from the coordinates of the ends of the lines. A line may represent more than one member. For example, where the chord of a truss is represented in the drawing as a single line, Microstran will detect that other members intersect with it, or that the ends of other members lie on it and will sub-divide it automatically into the series of members required for analysis.

On selecting the **File > Import > CAD DXF** command a file selection dialog box is displayed so you can browse for the DXF file that you wish to import. The dialog box below is then displayed.



SELECT DXF LAYERS

Layers

All layers in the DXF are shown in the list box and you can select any that contain lines to be imported. Only lines in the specified layers will be imported. Microstran assigns each specified layer a number and every member found on that layer is assigned a section number equal to the layer number.

Members are numbered in the order in which they are present in the DXF. All members from a sub-divided line are grouped together. Nodes are numbered in order of increasing vertical axis coordinate.

Factor

A coordinate conversion factor – the multiplication factor to be applied to drawing coordinates to produce coordinates used in the analysis. If, for example, the drawing is in mm and the Microstran model is to be in meters, the factor would be 0.001.

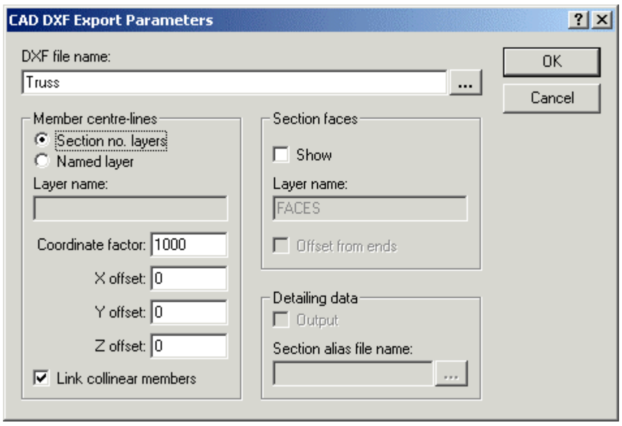
Tolerance

The minimum node separation. Nodes separated by less than this distance will be treated as coincident.

Exporting a CAD DXF

Member centre-lines are represented by a single line in the DXF. If section properties have been extracted from a steel section library the section shape may also be represented by a number of planes. The section shapes may be curtailed at member ends to avoid overplotting at the intersections.

On selecting the **File > Export > CAD DXF** command the dialog box below is displayed.



CAD DXF EXPORT PARAMETERS

Do not check the box for detailing output unless you are using a Microstran-compatible detailing interface.

The DXF contains only an Entities section without a drawing header. After reading the file into AutoCAD with the "DXFIN" command, you may use the "ZOOM E" command to fill the screen with the drawing. The limits may then be adjusted as required.

You may suppress hidden lines and render the drawing using AutoCAD.

Windows Clipboard Operations

Microstran facilitates use of the Windows clipboard for transfer of images to CAD programs by using the Enhanced Metafile Format (EMF) for the Windows clipboard when you select the **View > Copy** command. In programs such as AutoCAD, you can use the **Paste** command to directly insert an image of the main Microstran view. Pressing the Print Screen key on the keyboard writes a Windows bitmap to the clipboard. Both of these formats may be pasted into Microsoft Word documents.

14:Analysis

General

Microstran offers a number of static and dynamic analysis options, each of which employs exhaustive *consistency checking* and highly efficient equation solution procedures.

Linear Elastic Analysis is a first-order elastic static analysis in which non-linear effects are ignored and the stiffness equations are solved for only the primary load cases. Solutions for combination load cases are obtained by superposition of the solutions for the primary load cases.

Non-Linear Analysis is a second-order elastic analysis, which enables you to take into account the non-linear actions arising from the displacement of loads (the P-Δ effect), the change in flexural stiffness of members subjected to axial forces (the P-δ effect), and the shortening of members subjected to bending (the flexural shortening effect). Non-linear analysis is an iterative procedure in which the behaviour at each step is controlled by a number of parameters. Each selected case, whether a primary or combination load case, is solved separately because superposition of results is not valid. Members defined as *tension-only*, *compression-only*, *gap*, or *brittle fuse* are checked at each iteration and included or excluded according to their respective constitutive rules.

Elastic Critical Load Analysis calculates the frame buckling load factor, λ_c, for selected load cases and computes the corresponding member effective lengths for each load case. λ_c is used in steel design codes for the computation of moment amplification factors.

Dynamic Analysis computes the natural vibration frequencies of the structure and the associated mode shapes. The dynamic loads on the structure due to earthquake or other support acceleration may then be assessed using the *response spectrum* method.

The **Master-Slave Constraints** option is available with all types of analysis and is particularly useful for the modelling of elements that are effectively rigid in their own plane, such as floor slabs.

The **Profile Optimizer** is used in all analyses to minimize analysis time and storage requirements. Nodes and members can be numbered for maximum convenience in data generation and interpretation of results.

Method

Microstran uses the well-documented direct stiffness method of analysis in which the *global stiffness matrix*, [K], is assembled from the stiffness contributions of individual members. For large structures, [K] can be quite large and is stored on disk in blocks sized to maximize the use of available memory and to minimize solution time. Load vectors, **P**, are formed from the applied loads and node displacements, **u**, are determined by solving the equation:

P = [K] u

The forces in each member are then determined by multiplying the member stiffness matrix by the appropriate terms of the displacement vector, resolved into member axes.

Consistency Check

Microstran performs an automatic check of all input data prior to analysis. The consistency check will detect a range of modelling problems related to geometry and loading. Data errors and warnings are shown in the output window and are also written to the error report, which can be listed and printed using options on the File menu.

Accuracy

All analyses use double-precision arithmetic to minimize the loss of precision inherent in the many arithmetic operations required for solving large, complex structural models. Microstran analysis modules report the *condition number* (CN) and maximum *residuals* to assist in interpretation of the analysis results.

Condition Number

The condition number, a measure of the loss of precision that has occurred during solution, is calculated after the decomposition of the **[K]** matrix. A "well-conditioned" structural model is one in which the condition number is less than about 10^4. If the condition number exceeds this value you should treat the results with caution and look for evidence of "ill-conditioning". For example, the large displacement of a node or group of nodes may indicate that the structure or part of it is acting as a mechanism. See "Report Contents".

For Non-linear analysis the condition number is computed only after the first analysis iteration and may not give a meaningful measure of the final analysis result. If displacement control is actuated the condition number reflects the transient stabilization of the model and is unreliable.

Residuals

An important independent check on the accuracy of the solution is provided by the node equilibrium check. At unrestrained nodes the sum of all the member end actions is compared to the sum of external forces

acting on the node. Any difference is a *force residual*, the out-of-balance force. The maximum residual is reported in the Analysis window. The maximum residual should be considered in conjunction with the magnitudes of the applied loads in assessing the adequacy of the solution.

Note: A satisfactory equilibrium check, by itself, is not sufficient to ensure an accurate solution – the condition number must also be satisfactory. See "Instability & Ill-Conditioning" on page 72.

Linear Elastic Analysis

Linear elastic analysis cannot be performed if there are any non-linear members in the model (tension-only, compression-only, cable, gap, or fuse). An error message will be displayed if you attempt linear analysis of a model containing non-linear members.

All load cases are analysed when you choose linear analysis. Results for combination load cases are determined by superposition of the results of the component primary load cases.

Note: If you perform a non-linear analysis and then a linear analysis, the settings in the Select Analysis Type dialog box will be lost (see "Selecting Load Cases for Non-Linear Analysis" on page 251). Performing a linear analysis sets the analysis type flag to **L** (linear).

Non-Linear Analysis

Non-Linear Analysis (also called second-order analysis) performs an elastic analysis in which second-order effects may be considered. The different second-order effects are described below.

Non-linear analysis uses a multi-step procedure that commences with a linear elastic analysis. The load residuals, computed for the structure in its displaced position and with the stiffness of members modified, are applied as a new load vector to compute corrections to the initial solution. Further corrections are computed until convergence occurs.

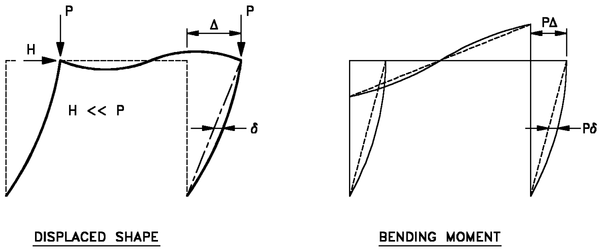
There is no single method of iterative non-linear analysis for which convergence is guaranteed. It may therefore be necessary to adjust the analysis control parameters in order to obtain a satisfactory solution.

The solution may not converge if the structure is subject to gross deformation or if it is highly non-linear. This may be the case as the elastic critical load is approached.

Note: You should not attempt to use non-linear analysis to determine elastic critical loads. Results of non-linear analysis should be treated with caution whenever the loading is close to the elastic critical load.

Second-Order Effects

The most important second-order effects taken into account in non-linear analysis are the *P-Delta effect (P-Δ)* and the *P-delta effect (P-δ)*. These are discussed in detail below.



P-Δ AND P-δ EFFECTS

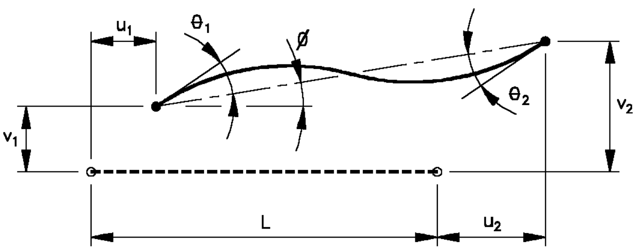
You may independently include or exclude these two major effects. Different combinations of the P-Δ and P-δ settings affect the operation of non-linear analysis as set out in the table below.

Node Coordinate Update *	Axial Force Effects	Analysis Type
NO	NO	Linear elastic analysis with tension-only or compression-only members taken into account.
YES	NO	Analysis includes the effects of displacement due to sidesway but not changes in member flexural stiffness due to axial force. These settings will usually yield satisfactory results for pin-jointed structures.
NO	YES	Full account is taken of the effects of axial force on member flexural stiffness while the effects of node displacement are approximated by a sidesway correction in the stability function formulation. These settings normally give minimum solution time with second-order effects taken into account.
YES	YES	This is the default analysis type, which provides the most rigorous solution for all structure types.

* Node coordinate update is automatically enabled for models containing cables.

Node Coordinate Update – P-Delta Effect

The *P-Delta effect (P-Δ)* occurs when deflections result in displacement of loads, causing additional bending moments that are not computed in linear analysis. P-Δ is taken into account either by adding displacement components to node coordinates during analysis or by adding sidesway terms to the stability functions used to modify the flexural terms in the member stiffness matrices. Either *small displacement theory* or *finite displacement theory* may be used with node coordinate update. As shown in the diagram below, finite displacement theory takes into account the rotation of the chord of the displaced member in computing the end rotations and the extension of the member. Only where large displacements occur would the use of finite displacement theory produce results different from those obtained with small displacement theory.



SMALL DISPLACEMENT THEORY

Axial extension: $u_2 - u_1$

End rotations: θ_1, θ_2

FINITE DISPLACEMENT THEORY

Axial extension: $\sqrt{(L+u_2-u_1)^2 + (v_2-v_1)^2} - L$

End rotations: $\theta_1 + \emptyset, \ \theta_2 + \emptyset$

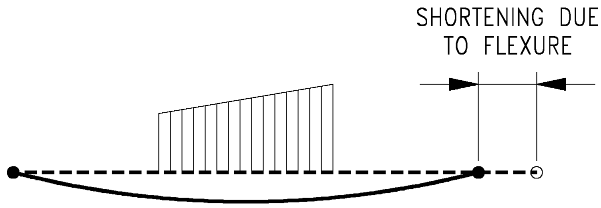
SMALL AND FINITE DISPLACEMENT THEORIES

Axial Force Effects – P-delta Effect

The bending stiffness of a member is reduced by axial compression and increased by axial tension. This is called the *P-delta effect (P-δ)* and is taken into account by adding beam-column stability functions to the flexural terms of the member stiffness matrices. Member stiffness matrices therefore vary with the axial load and are recomputed at every analysis iteration. The stability functions are derived from the “exact” solution of the differential equation describing the behaviour of a beam-column. The additional moments caused by P-δ are approximated in some design codes by the use of moment magnification factors applied to the results of a linear elastic analysis.

Flexural Shortening

As a member flexes under the action of bending moments its chord shortens, giving rise to an apparent decrease in axial stiffness. This effect, known as the flexural shortening effect or the bowing effect, is shown in the diagram below. If the member is also subjected to axial load, the flexural shortening is increased by compression and decreased by tension. Some formulations for flexural shortening consider only the effects of end moments but Microstran also takes account of any loads applied along the member.



FLEXURAL SHORTENING

Changes in Fixed-End Actions

Member fixed-end actions may change between successive analysis iterations owing to displacement of the member and variations in its flexural stiffness caused by axial force. Microstran automatically recalculates the fixed-end actions at each analysis iteration and updates the load vector accordingly.

Non-Linear Members

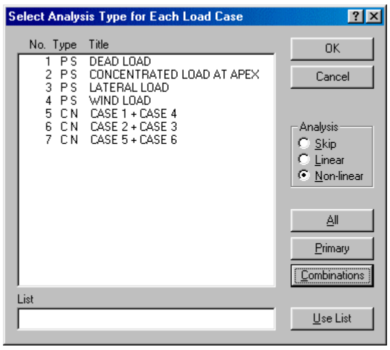
Analysis of models containing non-linear members requires non-linear analysis, irrespective of whether second-order effects are to be considered. At the conclusion of each analysis step, all tension-only, compression-only, gap or fuse members are checked and may be removed from or restored to the model for the next analysis step depending on the relative displacement of end nodes. If the removal of non-linear members causes the structure to become unstable, no solution is possible.

Running a Non-Linear Analysis

Selecting Load Cases for Non-Linear Analysis

Non-linear analysis lets you specify the load cases to be analysed and the analysis type to be used for each load case. A load vector is formed for each load case to be solved, whether it is a primary load case or a combination load case. *There is no need to analyse any load cases for which results are not required.*

On selecting the **Analyse > Non-Linear** command the dialog box shown below is displayed. In the Type column, load cases are identified as **P**rimary or **C**ombination. The second character is a code that specifies whether the load case is to be processed with **L**inear analysis or **N**on-linear analysis, or is to be ignored (**S**kipped). The linear setting allows you to perform iterative linear analysis on models with non-linear members. The non-linear setting is required by some design codes.



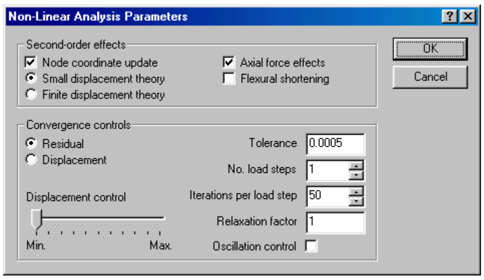
SELECTING LOAD CASES
FOR NON-LINEAR ANALYSIS

In general, only "realistic" load cases should be selected for non-linear analysis – there is no point in analysing a wind load case, for example, because this load will never exist in isolation. This is particularly important for structures containing cable elements because analysis may not be possible unless self-weight is included.

Note: The settings in this dialog box will be lost if you subsequently perform a *linear* analysis. In this case, the analysis type flag (S/L/N) will be unconditionally set to **L**inear. You must reinstate the analysis type flag if you revert to non-linear analysis.

Non-Linear Analysis Parameters

The dialog box shown below determines the characteristics of the non-linear analysis.



NON-LINEAR ANALYSIS PARAMETERS

The dialog box contains the following items:

- **Node coordinate update (P-Δ)**
  This flag is set if node coordinates are to be updated at each analysis step. It is automatically set for structures containing cable elements. The default setting is on.
- **Small/finite displacement theory**
  If the node coordinate update flag has been set, either small or finite displacement theory must be selected. Small displacement theory is the default setting.
- **Axial force effects (P-δ)**
  If this flag is set member stiffnesses are modified at each analysis step. The default setting is on.
- **Flexural shortening**
  This flag may be set only if the axial force effects flag is set. Set this flag to take into account the variation in member axial stiffness caused by the shortening of members subjected to flexure. The default setting is off.
- **Residual / displacement**
  Specifies the criterion to be used for convergence of the analysis. Residual uses a function of the maximum out-of-balance force after analysis. When Displacement is selected convergence is determined by comparing the convergence tolerance against a generalized measure of the change in displacement between successive iterations. For a satisfactory solution there must be acceptably small changes in the displacement and the residual must be small. The default setting is Residual.
- **Displacement control**
  Increasing the setting of this control may assist convergence in situations where displacements appear to diverge with successive analysis iterations, or for structures that are initially unstable but become stable as they displace under load. You should usually leave this control at minimum and only increase the setting if difficulties

are encountered in solution. Settings above minimum affect the calculation of the condition number.

- **Convergence tolerance**
  This value determines when the analysis has converged, determined by checking the change in the convergence criterion between successive analysis cycles. Too small a value will prolong the solution time and may even inhibit convergence. The default value of 0.0005 should be decreased if prescribed displacements have been used.
- **No. load steps**
  You may apply loads in a stepwise fashion which may assist in obtaining a solution for flexible structures by keeping displacements small at each load increment. This parameter is usually left at its default value of 1.
- **Iterations per load step**
  The maximum number of analysis iterations for each load step. This parameter is used to stop the analysis if convergence is taking an excessive time. The default value is 50, but larger values may be required for very flexible structures or those containing large numbers of cable elements.
- **Relaxation factor**
  The relaxation factor is applied to incremental displacement corrections during analysis. The optimum value for the relaxation factor depends on the type of the structure. As a general rule, structures which “soften” under load (i.e., displacements increase disproportionately with load) have an optimum relaxation factor between 1.0 and 1.2 while structures which “harden” under load have an optimum relaxation factor as low as 0.85. Caution is recommended in changing the relaxation factor from the default value of 1.0; if the relaxation factor is too far from optimum the analysis may require an excessive number of iterations for convergence or it may not converge at all.
- **Oscillation control**
  This control facilitates convergence when the solution oscillates owing to the removal and restoration of tension-only or compression-only members. The default setting is off.

As the analysis proceeds, the analysis window displays key information for each selected load case. At each analysis iteration the maximum values of residual and displacement are displayed. Note that at this stage, the values shown are for the most critical degree of freedom, i.e., residuals may be either forces or moments, and displacements may be either translations or rotations.

Instability

Instability detected during linear analysis is usually due to modelling problems and some of the common causes of these are discussed elsewhere.

Because a non-linear analysis considers the effects of axial force on member stiffness it is able to detect a range of instability that linear analysis cannot. For example, non-linear analysis may detect buckling of individual members or of the whole frame. The manner in which a structure is modelled and the analysis parameters used can have some bearing on the stage of the analysis when instability of individual members is detected and the way in which it is subsequently treated. If an unstable member is detected during the update process at the end of each iteration, it will be deleted from the following iteration. The presence of unstable members is reported in the Analysis window and details are written to the static log file. However, if the instability is not in a single member but localized in a small group of members it may not be detected until the completion of the analysis. In this case, the presence of the instability will be reported in the Analysis window and some diagnostic information will be written to the static log file to assist you in correcting the problem. Even though the analysis has failed, results are available and may be used to determine corrective measures, e.g. increase some member sizes or, perhaps, change to tension-only members. *The results of an analysis in which instability has been reported are useful for diagnosis but should not be used for other purposes.*

An elastic critical load analysis will often assist in locating the cause of local instabilities.

Troubleshooting Non-Linear Analysis

It is possible to perform a successful *linear analysis* for structures that are incapable of resisting the imposed loads. *Non-linear analysis* is a more complete simulation of the behaviour of a structure under load and the procedure may fail to provide a solution where a linear analysis succeeds. This may occur, for example, if some compression members are slender and buckle. When non-linear analysis fails to converge, the following tips may be helpful:

- Make sure that a linear analysis can be performed. If not, troubleshoot the linear analysis before continuing with the non-linear analysis.
- Is a full non-linear analysis necessary? If the only significant non-linear effect is the presence of tension-only or compression-only members, set the analysis type to **L** for these load cases. In other cases, a successful analysis may result if either node coordinate update or axial force effects are excluded.
- Examine the static analysis log file. It contains information about members that have buckled or become ineffective.

- Perform an elastic critical load analysis to check the frame buckling load. If it is less than the imposed load non-linear analysis is not possible.
- Is the structure too flexible? Remove excessive member end releases (pins). Sometimes, in diagnosing convergence problems, it is helpful to remove ALL releases and reinstate them in stages.
- Adjust non-linear analysis parameters.

Elastic Critical Load Analysis

Elastic Critical Load (ECL) Analysis performs a rational buckling analysis to compute elastic critical loads for the frame and the associated buckling mode shapes. Member effective lengths are determined.

The elastic critical load provides a practical, reasonable estimate of the collapse load of a rigid-jointed structure (Livesley). The elastic critical load of a structure is essentially a generalization of the Euler buckling load of a pin-ended strut, at which the displaced position of the structure is maintained without any additional load and the equilibrium configuration of the frame is not unique.

Note: Lateral torsional buckling (also called flexural-torsional buckling) is not taken into account in ECL analysis.

The elastic critical load of the structure is a function of the elastic properties of the structure *and the applied loading.* The elastic critical load is computed separately for each load case and is reported as a buckling load factor *(λc),* which is the factor by which the initial axial forces in the frame must be multiplied to cause the structure to become unstable. A load factor of less than 1.0 for any load case indicates that the structure is unstable under the applied loading. Normally, only the first buckling mode is required but higher modes will be of interest if lower modes represent localized buckling behaviour or they are inhibited by constraints not in the model.

The initial axial forces in a frame are obtained from a static analysis. A linear elastic analysis is usually sufficient but non-linear analysis must be used when the structure contains non-linear members.

Restraints affecting the flexural buckling behaviour of the structure must be included in the structural model. For example, if out-of-plane buckling behaviour is to be considered for a plane frame, the frame must be modelled as a space frame with the appropriate out-of-plane restraints.

Buckling mode shapes can be displayed for each load case. The mode shape indicates whether buckling involves the frame as a whole or just localized buckling of one or more frame members.

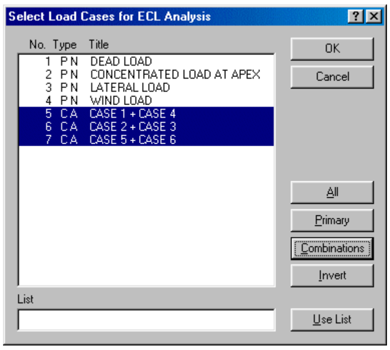
The *effective length* of a compression member is the length of an equivalent pin-ended member whose Euler load equals the axial force in

the member when the frame buckles. Effective length factors, which are multiplying factors (k) applied to the actual member length, are calculated for both member principal axes for each load case.

Elastic critical load analysis is not recommended for structures containing cable elements because of the highly non-linear nature of structures of this type.

Selecting Load Cases for ECL Analysis

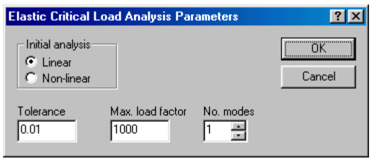
Select the **Analyse** > **Elastic Critical Load** command. The dialog box below is displayed for you to select the required load cases. Usually you would select only combination load cases for design.



SELECTING LOAD CASES FOR ECL ANALYSIS

Analysis Control Parameters

The settings in this dialog box determine the type of elastic critical load analysis that will be performed.



ECL ANALYSIS PARAMETERS

The dialog box contains the following items:

- **Initial analysis**
  Sets the initial analysis type to determine the distribution of axial forces used for the elastic critical load analysis. It is normally **Linear** but must be **Non-linear** if the structure contains non-linear members.

- **Tolerance**
  The tolerance is the relative accuracy to which load factors are required. Too small a value will prolong the solution time. The default value is 0.01.

- **Max. load factor**
  The search for the elastic critical load will terminate if the load factor exceeds this limiting value. The default value is 1000.
- **No. modes**
  The number of buckling modes to be computed for each selected load case. Normally, only the first mode is required. Higher modes will be of interest if lower modes represent localized buckling behaviour.

When the analysis is finished a summary of results appears in the analysis window. The summary shows for each selected load case the critical load factor and the most critical member with associated k values.

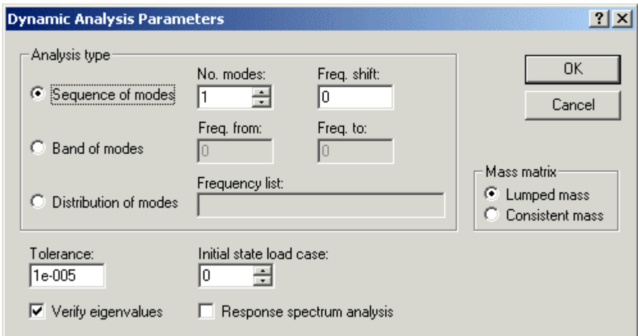
Dynamic Analysis

Dynamic Analysis computes the frequencies and mode shapes of the natural vibration modes of the structural model. Only the mass and stiffness of the model are considered in computing natural frequencies and mode shapes. Static load cases are ignored. The frame mass is computed automatically and additional mass that is to be taken into account may be modelled as node masses. Member masses are computed automatically as the product of the cross-sectional area and the mass density. Additional node masses may be input as required. The unit used for mass must be consistent with the force and length units.

Select the **Analyse > Dynamic** command to start dynamic analysis.

Analysis Control Parameters

After selecting load cases, the dialog box shown below appears. The settings in this dialog box determine the type of dynamic analysis that will be performed.



DYNAMIC ANALYSIS PARAMETERS

The dialog box contains the following items:

- **Sequence of modes**
  This is the default method – modes will be computed from the lowest frequency.
- **No. modes**
  The number of natural frequencies and mode shapes that can be computed is limited by the number of dynamic degrees of freedom, and, for large structures, by the amount of available memory. Solving for a large number of modes is usually not warranted.
- **Freq. shift**
  Modes will be found above this frequency value (Hz).
- **Band of Modes**
  In this method only modes lying between the from and to values will be computed (Hz).

- **Freq. from**
  Modes with a frequency less than this value (Hz) will not be found.
- **Freq. to**
  Modes with a frequency greater than this value (Hz) will not be found.
- **Distribution of modes**
  Use this method to count the number of modes whose frequencies are lower than the specified values (Hz). Mode shape results are not produced.
- **Frequency list**
  Modes with a frequency between the first and last values (Hz) will be found.
- **Tolerance**
  This is the tolerance to be used in determining the convergence of eigenvalues. If the value is too small, convergence may not be possible or an excessive number of iterations may be required. If the value is too large, the eigenvalues found may not be the lowest. The default value is 0.00001.
- **Verify eigenvalues**
  Check this box if you wish to verify that no eigenvalues have been skipped in the computation (see above).
- **Lumped mass / Consistent mass**
  The mass matrix may be computed using either a lumped mass or consistent mass or formulation. The latter is usually only required for beam models without nodes between supports.
- **Initial state load case**
  Non-linear behaviour is not taken into account in dynamic analysis but it is possible to specify a load case that defines the initial state. For example, a leeward cable in a guyed mast subjected to wind load may be slack. If the corresponding load case is specified as the initial state load case, the slack cable will be eliminated from the analysis. The default value is zero.

Response spectrum analysis
You must check this box if you wish to proceed to a response spectrum analysis after the dynamic analysis.

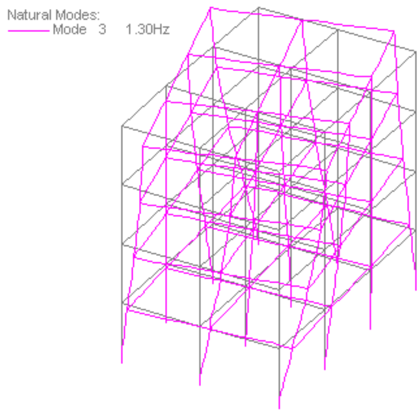
Dynamic Modes

After completing a dynamic analysis it is important to check the mode shapes to ensure that you have the required dynamic modes. Microstran computes all dynamic modes, including torsional modes. The easiest way to examine the results is to display an animated view of the computed mode shapes.

Note: You can add low-mass "semaphore" members to visualize torsional modes.

Dynamic Analysis Example

Example EX6 is a rectangular building frame, 3 bays x 2 bays x 4 storeys, which was generated with Standard Structures Input. Additional mass of 10 tonnes is attached at each unrestrained node. Dynamic analysis shows the first two modes are sidesway modes and the third is a torsional mode. The mode shape associated with the third mode is shown below.



NATURAL MODE SHAPE

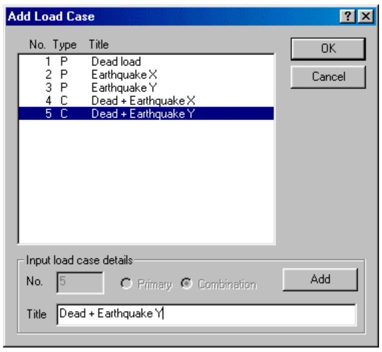
Response Spectrum Analysis

Response Spectrum Analysis is used to determine peak displacements and member forces due to support accelerations.

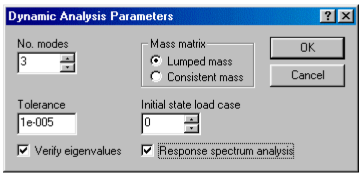
Running a Response Spectrum Analysis

The procedure for performing a response spectrum analysis is:

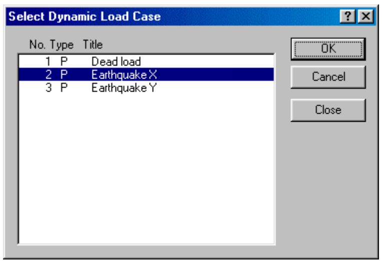
1. Set up static analysis load cases using the dialog box shown below and perform the static analysis. The earthquake load cases are empty – results from the response spectrum analysis will be added automatically.



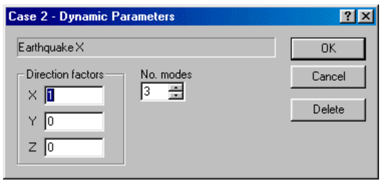
2. Select dynamic analysis and check item **Response spectrum analysis**.



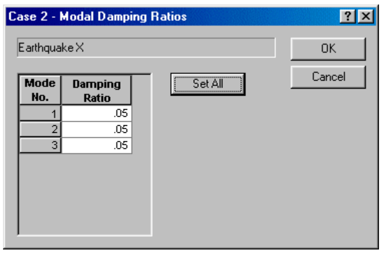
3. You are next prompted to identify the load cases that are to be used for the results of the response spectrum analysis. There will be one such load case for each earthquake direction being considered.



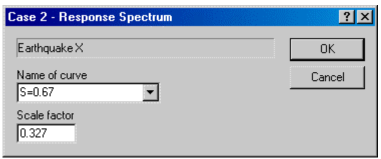
4. For each earthquake load case you must enter parameters to determine the response spectrum direction and the number of modes to be considered. The direction factors determine the direction of the support acceleration in terms of components in the global axis directions. These components will be reduced to a unit vector before being used. The number of modes must be sufficient to satisfy the earthquake code requirement that 90% (typically) of the seismic mass is accounted for. It must not be greater than the number of modes computed during dynamic analysis (2, above). The Delete button removes all dynamic analysis data associated with the load case.



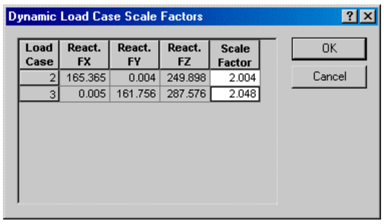
5. For each earthquake load case damping ratios are specified. The "Complete Quadratic Combination" method (CQC) for combining modal responses is used to determine the peak response. This is equivalent to the "Square Root of the Sum of Squares" (SRSS) method if all modal damping ratios are zero.



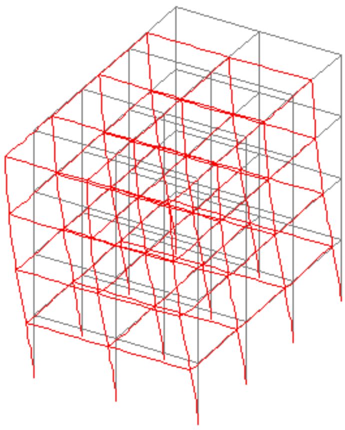
6. For each earthquake load case a response spectrum curve and scaling factor must be specified. The response spectrum curve is chosen from a list of names of digitized response spectrum curves contained in the file *Response.txt*. The scale factor is used to multiply the spectral acceleration values to give the actual support acceleration to be used in the analysis. You may edit the response spectrum curves or add new ones using the **Configure > Edit Response Spectra** command.



7. After steps 3-6 have been completed for each earthquake case, the dynamic analysis proceeds. On completion, select the **Analyse > Response Spectrum** command to scale the computed actions and combine them with the static analysis results (note that this item is greyed out on the menu until all the necessary preconditions for response spectrum analysis have been completed). The total reactions (base shears) are displayed for each earthquake case and you may now enter scale factors determined by code requirements.



Microstran now adds the results from the response spectrum analysis to the static analysis results. Earthquake load cases may now be treated as any other load case for the display and reporting of results and for design. The diagram below shows the displaced shape for one of the earthquake load cases.



DISPLACED SHAPE FOR EARTHQUAKE LOAD CASE

Note: The displaced shape represents the **peak values** of the displacement during the earthquake event. There are no negative values. Interpretation of the results should take this into account.

Response Spectrum Scale Factor

The results of the static analysis are updated with the results of the response spectrum analysis. As this process takes place, the sum of the reactions for each dynamic load case will be displayed and you may enter factors that will be used to scale the results to ensure compliance with codes that require minimum base shears (step 7, above). The factor should be based on the base shear in the direction of the support acceleration. Note that the values given for the reactions are the sum of absolute values, as the methods used to combine individual modal responses result in loss of sign.

The results for each dynamic load case are inserted in the results files for the previously defined empty load cases. Any combination case that refers to the dynamic case is updated by adding the specified dynamic case, factored as specified. By updating combination cases instead of computing them completely from the results of primary cases, any non-linearity in the previously computed results is preserved. However, the static analysis must be repeated if the dynamic analysis is to be amended.

Note: After running response spectrum analysis you should look at the dynamic analysis log file, which contains important data including mass participation factors.

Earthquake Design Code Spreadsheets

The Excel spreadsheets **AS1170_4.xls** and **Nzs1170_5.xls** set out detailed procedures for performing response spectrum analysis complying with the design codes AS 1170.4 and NZS 1170.5, respectively.

Response Spectrum Curves

The digitized data for the response spectrum curves must be entered into the **Response.txt** file, which resides in the library folder. This is a text file that you may edit to add additional response spectrum data. The format of each set of data in the file is as follows:

```
Name
T(1)   Sa(1)
T(2)   Sa(2)
T(3)   Sa(3)
.....
T(n)   Sa(n)
END
```

where:

`Name`	String of alphanumeric characters used to identify each curve.
`T(n)`	Period in seconds for the nth point on the curve.
`Sa(n)`	Spectral acceleration for the nth point on the curve. The spectral accelerations may be in normalized form or as absolute accelerations with a scale factor, described previously, being used to effect any required conversion.
`END`	Keyword, indicating the end of data for this curve.

The data in **Response.txt** is available in graphical form in an Excel spreadsheet, **Spectra.xls**, also in the library folder.

Errors

There are some types of error that only become evident during analysis and it is not possible for the consistency check to warn of this type of error before the analysis commences. For example, if a structure is unstable because some part of it actually forms a mechanism, analysis will be terminated and an error message will be displayed on the screen. The error message is of the form:

```
STRUCTURE UNSTABLE AT NODE nnnnn DOF f
```

where:

`nnnnn`	=	The node number at which instability was detected.
`f`	=	The DOF number, as shown in the table below, in which there was found to be no resistance to displacement.

Sometimes in linear elastic analysis a modelling problem may manifest itself as gross linear or angular displacement. This kind of problem may not be obvious in the member force plots but may be evident in the plot of displaced shape. Modelling problems of this type can usually be fixed by the addition of one or more node restraints to inhibit the gross displacement.

In non-linear analysis very large displacements can occur in the analysis of structures containing very flexible tension members. If displacements are sufficiently large the analysis will be terminated with a message of the form:

```
EXCESSIVE DISPLACEMENTS
```

A solution can sometimes be obtained in cases like this by adjusting the analysis parameters but it is preferable to model very flexible tension members as *cables*.

The above error message may also be obtained where the automatic deletion of tension-only bracing members during non-linear analysis renders a structure unstable. See “Tension-Only & Compression-Only Members” on page 97 and also “Instability & Ill-Conditioning” on page 72.

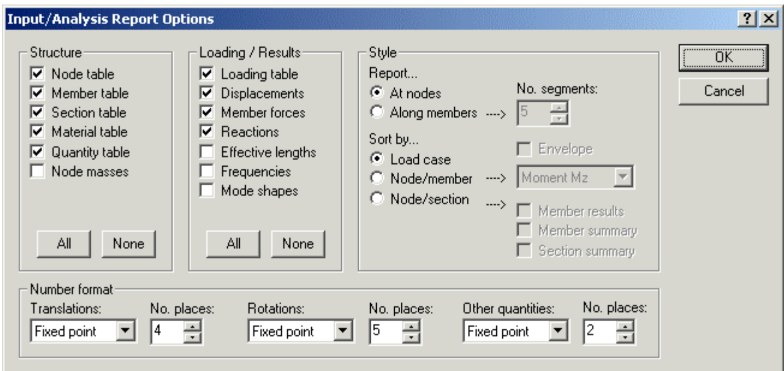
15:Reports

General

Having selected **Reports** on the menu bar you may choose an **Input/Analysis** report, a **Steel Design**, or an **RC Design** report. Tables of structure data, loads, or results may be limited to the specific nodes, members, and load cases required. This ability to select exactly what you want is particularly useful in reporting for large structures. Reports are available at any stage of a job – tables of structure data may be produced prior to analysis. Several sample reports are shown in Chapter 20 – "Examples".

Input/Analysis Report Options

The report options dialog box is displayed when an Input/Analysis report is requested and this allows you to specify the tables required in the report. The number format may be specified separately for translations, rotations, and other quantities.



INPUT/ANALYSIS REPORT OPTIONS

The options in the Style group determine the report type:

Report *At nodes* or *Along members*

You may report member results either at the nodes only or along the members (including the end nodes). Only when you select results along members can you choose the number of segments in each member. By default, there are 5 segments in each member, giving 7 values for each result – at the end nodes and at 5 intermediate locations. There are additional intermediate points when there are concentrated loads.

Sort by...

Load case

The selected loading/results will be sorted by node/member number within load case number. With this setting you cannot select *envelopes*.

Node/member

The selected loading/results will be sorted by load case number within node/member number. This is the only setting for which envelopes are possible. If you check the Envelope check box you may then choose the member force component (e.g. bending moment Mz) for which maxima and minima are to be reported.

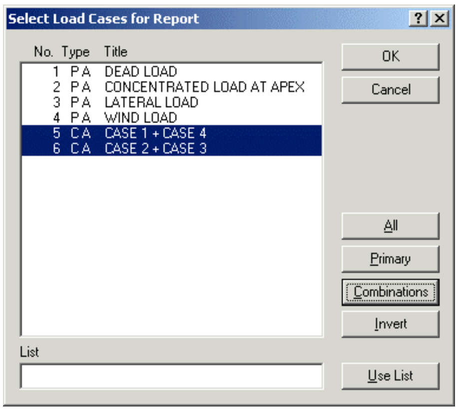
Node/section

With this setting node results are sorted by load case within node number and member results are sorted by load case within *section* number. These reports may be very useful for design. The adjacent check boxes allow you to independently include the member results in three different formats:

- **Member results**
  For each member, in section number order, member forces (and optionally displacements) for each load case.
- **Member summary (envelope)**
  For each section, a summary of minimum and maximum member forces. With each minimum and maximum member force the member number and load case are shown, together with the co-existing member forces.
- **Section summary (envelope)**
  For each member, a summary of minimum and maximum member forces. With each minimum and maximum member force the load case is shown, together with the co-existing member forces.

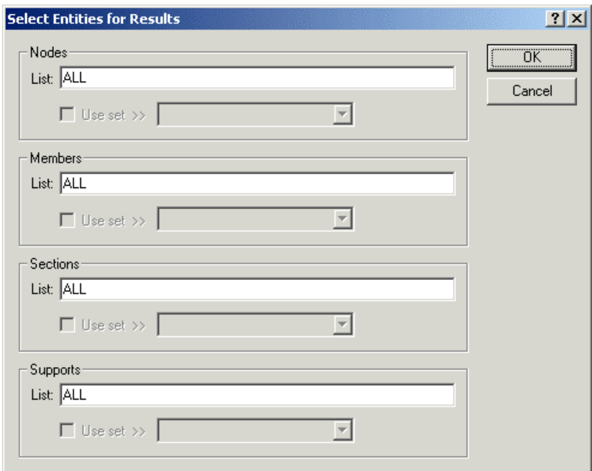
Limiting the Scope of the Report

You may limit the scope of the loading/results tables included in the report by selecting load cases and entities (nodes, members, and sections). When loading or results are included in the report options you must specify the load cases in the dialog box shown below. If a load case selection is in effect, this will be the default selection for the dialog box. You may specify a list of load cases by entering a list and clicking the Use List button.



SELECTING LOAD CASES FOR REPORT

For each table of loading or results selected in the report options dialog box you may specify the entities for which reporting is required. In addition to a list of entities you may also specify a set containing these entities. By default, all items are selected with the ALL keyword. Edit boxes are disabled if results were not requested for the corresponding entity.



ENTERING REPORT LISTS

In entering a list you may use spaces or commas for separators. A hyphen is used to specify a range. You may also use any legitimate combination of the keywords ALL, TO, INC, and X. Some examples of lists are shown in the table below:

List	Meaning
1,3,5,6	Items 1, 3, 5, and 6
9-15	Items 9 to 15 inclusive
9-	All items from 9 onwards
-9	All items up to 9 inclusive
5 TO 50 INC 5	Items 5, 10, 15, 20, ... 45, 50
ALL X 5 TO 10	All items excluding 5 to 10 inclusive

Report Contents

All pages record the name of the licensed user, the job name, the job title (two lines), the page number, the total number of pages, the date and time of printing, the program version, and the full path of the job.

Title Page

A job summary on the title page contains the following information:

- Job name.
- Title.
- Structure type.
- Date and time of report.
- Number of nodes.
- Number of members.
- Number of spring supports.
- Number of sections.
- Number of materials.
- Number of primary load cases.
- Number of combination load cases.
- Analysis type.
  For non-linear analysis the values of the analysis control parameters are also shown (see “Non-Linear Analysis” on page 247).
- Load case summary.
  This lists titles of all load cases reported, together with the load case type and a flag to indicate the type of analysis performed (or whether analysis was skipped).

Structure Data

Structure data is reported in the following tables:

- **Node Coordinates**
  This table lists the global coordinates and the restraint code of all nodes in the structure. (A “1” in the restraint code means that a restraint exists at the corresponding DOF). See “Fixed Supports (Node Restraints)” on page 87.

- **Member Definition**
  This table lists, for all members, the member connectivity, the orientation, the section number, the material number, the release codes and the computed length. (A “1” in the release code means that a release exists in the corresponding direction). See “Member Definition” on page 83.

- **Library Sections**
  For each section extracted from a library, this table lists the library name, the section name, the orientation of the section (see “Section Properties” on page 88), and the comment field.

- **Standard Shapes**
  For sections input by specifying a standard shape, this table lists the section shape, name, comment field, and the defining dimensions.

- **Sections Input by Property Values**
  For sections input by specifying geometric property values, this table lists the section name and the comment field.

- **Section Properties**
  This table lists the geometric properties of all sections defined in the input.

- **Material Properties**
  This table lists the properties of all materials defined in the input.

- **Table of Quantities**
  The total length and mass are listed for each different section and for each different material. Normally, the values shown in the structure data tables will be identical to the input values but in some cases, there may be a small discrepancy because of round-off in the fixed format of the report table.

Condition Number

The condition number (CN) calculated during analysis is reported to give a measure of the degree of numerical accuracy that may be expected of the results. A large condition number is indicative of loss of numerical precision caused by modelling error. It is not possible to give a precise maximum allowable CN – some models that produce a CN of 10^4 are perfectly satisfactory, while others are not. As a rough guide, you should look carefully for signs of ill-conditioning if the CN approaches or exceeds 10^4.

The node number and the DOF at which the maximum CN occurs are reported to assist in the diagnosis of any problem. See “Instability & Ill-Conditioning” on page 72 and Chapter 14 – “Analysis”.

Applied Loads

For primary load cases applied loads are listed in the following groups:

- GRAV loads
- Node loads
- Node temperatures
- Member loads
- Member temperatures

For combination load cases the factors and titles of the component load cases are listed.

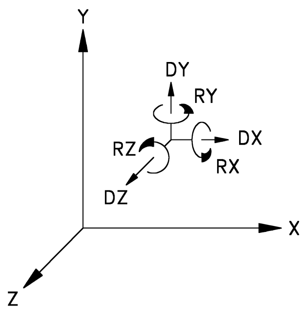
The sum of the applied loads acting in each of the global axis directions is reported for each load case.

Node Displacement Table

Node displacements are reported if selected in the report options dialog box and if results are being reported at nodes; i.e., node displacements will not appear if results are reported along members.

Microstran will request a list of nodes where displacements are to be reported. All nodes may be included (default) or lists of those to be included may be entered as set out above.

The displacements for each translational and rotational DOF are reported for each node specified in the node list. Displacements are measured with respect to the global axes and are positive in the positive axis directions. The diagram below shows positive displacement components.



POSITIVE DISPLACEMENT COMPONENTS

Large values for some displacements may indicate ill-conditioning, usually associated with excessive load residuals.

Member Force Tables

Member forces are listed if selected in the report options dialog box. The members for which member forces are reported may be selected as set out above.

The sign convention for member forces is such that positive member forces cause the following strains or displacements in the member:

- **Axial Force (F*x*)**
  Tensile strain.
- **Shear Force (F*y*, F*z*)**
  End "B" of the member deflects in the positive *y* or *z* direction relative to end "A".
- **Torque (M*x*)**
  End "B" of the member rotates in a positive sense relative to end "A".
- **Bending Moment (M*y*, M*z*)**
  Tensile strain occurs on the underside of the member (with respect to the member axes). That is, the member tends to sag.

See "Sign Conventions" on page 64 for a diagram showing the directions and the effects of positive member forces.

Member Force Envelope

Member force envelopes may be selected in the report options dialog box. In this case, it is necessary to specify the member force component and Microstran tabulates the minimum and maximum values of the specified component for the selected load cases.

Report at Nodes

Member forces acting on the ends of the selected members will be tabulated for each selected load case if reporting of results at the nodes was selected in the report options dialog box.

Report Along Members

Member forces and displacements will be tabulated for each member selected on the member list if reporting of results along members was selected in the report options dialog box. Only selected load cases will be reported. The number of segments to be used for calculating the intermediate values is also selected in the report options dialog box.

For each member, the "A" and "B" node numbers and the section number are listed. At the end of each segment in the member, the offset from the "A" end is listed, followed by all force components at that position.

Displacement components are tabulated as:

- **Global Components**
  Parallel to the global X, Y, and Z axes. These are absolute displacements.

- **Local Components**
  Parallel to the transverse member axes *y* and *z*. These components are reported relative to the chord (the line joining the displaced end nodes of the member). For cables sag is tabulated instead of these two components. Sag is measured in the direction of the load resultant on the cable.

Reactions Table

Reactions will be tabulated at support nodes if selected in the report options dialog box. The support nodes at which reactions are shown are listed as set out above.

The table of reactions gives *the forces acting on the structure* at the support nodes. These are the forces necessary to ensure the equilibrium of the structure. All forces are taken into account, including those applied directly to restrained DOF.

The sum of the reaction forces acting in each global axis direction are also reported. Each value should be equal to and opposite the sum of the applied loads in the direction of the respective axis.

Residuals Table

The maximum residual, or “out of balance” force, is reported for each primary load case when reactions are included in the report. The residual is obtained by subtracting the product of the structure stiffness matrix and the node displacement vector from the static equivalent of the applied loading at each node in the structure.

In all cases, the residuals should be small in comparison with the applied loading. Large residuals indicate that the structure is ill-conditioned.

Effective Lengths Table

For elastic critical load analysis, a table of critical loads and effective lengths is available. This table will be included if you selected effective lengths on the report options dialog box.

The elastic critical load factor is shown for the particular load case. This is the factor by which the input loads must be multiplied in order to obtain the elastic critical load of the structure for the load case. For each member, the following items are tabulated:

- **Length**
  Computed from end node coordinates.
- **Pcrit**
  The load in the member at the elastic critical load condition.
- **Pey**
  The Euler load for the member for buckling about the member *y* axis.

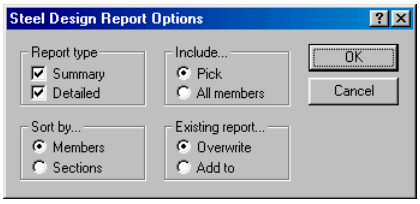
- ***ky***
  The effective length factor for the member for buckling about the member *y* axis.
- **Pez**
  The Euler load for the member for buckling about the member *z* axis.
- ***kz***
  The effective length factor for the member for buckling about the member *z* axis.

For more information, see "Elastic Critical Load Analysis" on page 255.

Design Report Options

Steel Design Report

This dialog box allows you to specify what kind of steel design report is required.



STEEL DESIGN REPORT OPTIONS

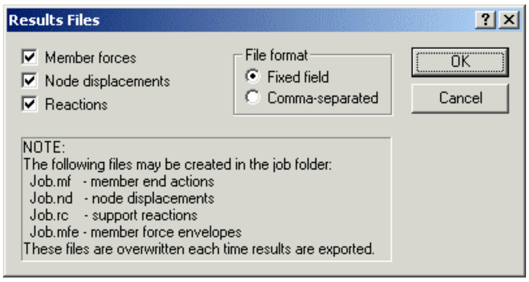
RC Design Report

RC design reports are produced automatically when RC design is run, so choosing the **Reports > RC Design** command will simply display a warning to this effect.

Exporting Results

Results files remain in the data folder after you close Microstran. You should occasionally review them in Explorer and delete any that are no longer required.

By selecting **File > Export > Results File** command, you may produce formatted text files of output results, which may be input to other programs or imported to spreadsheet programs, such as Excel. The files required are selected in the dialog box shown below. These files may be in either of two formats, fixed field or comma-separated. (User programs may create input data for Microstran by means of the archive file – see Chapter 9 – "Archive File Input".)



EXPORTING A RESULTS FILE

Numbers of nodes, members, load cases, etc. are formatted without a decimal place while real values may be in floating point or exponential format. The fixed field formats of the results files are as set out below.

Member Forces File

This file (Job.mf where "Job" represents the job name) contains the member force components at the ends of each member for each load case. The member forces are output with an internal sign convention where positive forces act in the positive member axis directions.

First line:

Number of members, number of load cases.

Following lines:

Member number, load case number, member end forces.

Node Displacements File

This file (Job.nd where "Job" represents the job name) contains the node displacements for each load case.

First line:

Number of nodes, number of load cases.

Following lines:

Node number, load case number, node displacements.

Reactions File

This file (Job.rc where "Job" represents the job name) contains the node reactions for each load case.

First line:

Number of restrained nodes, number of load cases.

Following lines:

Node number, load case number, reactions.

Member Force Envelopes File

This file (Job.mfe where "Job" represents the job name) contains the member force envelope for multiple load cases, for each member. The member forces are output with an internal sign convention where positive forces act in the positive member axis directions.

First line:

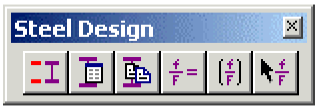
Number of members, number of load cases,
{ Fxmax | Fxmin | Fymax | Fymin | Fzmax | Fzmin |
Mxmax | Mxmin | Mymax | Mymin | Mzmax | Mzmin }.

Following lines:

Member number, load case number, member end forces.

16:Steel Member Design

General



STEEL DESIGN TOOLBAR

There are optional steel design modules that allow you to design and check steel members in a Microstran model in accordance with various steel design codes.

Most of the data required for a design is obtained directly from the model but additional information such as steel grade and cross-section restraints must be input. Once entered, this data is saved automatically when you save the Microstran job and when you export to an archive file.

The steps required prior to designing or checking members are:

- Analyse the structure.
- Initialize design members.
- Modify design data as required.
- Specify the load cases to be used in the design.

The **Design** > **Update Sections** command allows you to replace sections in the analysis model with those chosen from the design results. When this is done the analysis should be repeated to take account of any redistribution of member forces before a final design check.

Note: You must use kN and meter units in your Microstran model if you are going to use a steel design module.

Section Library

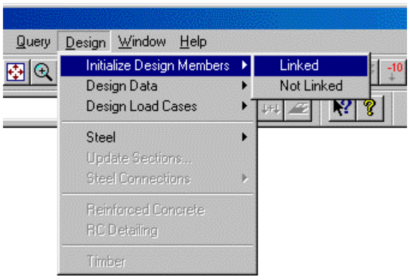
Design modules select sections from the configured steel section library. Library files containing sections used in Australia, Britain, Europe, Japan, New Zealand, and USA are available. Library files may be edited and special section libraries may be created. See Chapter 18 – “Section & Material Libraries”.

Analysis

The choice of linear or non-linear analysis should be based on design code requirements. If an elastic critical load analysis has been carried out you may choose to use automatically computed values for the effective lengths of compression members instead of specifying k factors. Additional moments caused by eccentric connections at the ends of channel, tee, and angle sections may be taken into account automatically in the design module.

Note: To express the adequacy of members, AS 4100 and NZS 3404 use the *load factor* (the ratio of capacity to load), which must be greater than or equal to 1.0. Other limit states design codes use the *utilization ratio*, which must be less than or equal to 1.0. Allowable stress codes use the *stress ratio* (the ratio of actual to allowable stress, f/F), which must be less than or equal to 1.0.

Initializing Design Members



DESIGN > INITIALIZE DESIGN MEMBERS

Initializing design members performs three necessary functions:

- Associates each initialized member with a specific steel design code.
- Assigns default design data to initialized members.
- Links analysis members together into *design members*, where appropriate.

Key concept.

In Microstran every design member that is not a cantilever has twist restraint at each end. Interior nodes that do not have twist restraint make it necessary to link analysis members together forming a design member. On initialization, no twist restraint is assumed at intermediate nodes in linked members.

The **Initialize Design Members > Linked** command initializes the selected members as linked design members if possible. Linked members must satisfy these conditions:

- They must be collinear (or almost so).
- They must have the same section number and orientation.
- They must have no releases.
- Their longitudinal axes must be in the same direction.

Member selected that do not satisfy these condition will be initialized unlinked.

Compression members that are not cantilevers must be linked so that design members span between positions of lateral restraint.

Key concept.

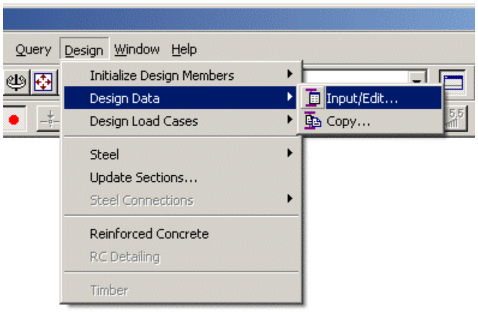
Use the **Initialize Design Members > Not Linked** command.. if you want each selected analysis member to be a design member. Twist restraint is then assumed at each end of each analysis member. *If this twist restraint does not exist unlinked initialization is not valid and you may obtain unconservative results.*

Key concept.

It is often convenient to select all the steel members in the structure that have to be designed and allow Microstran to link eligible members automatically. Use the **Query > Design Member** command to determine which analysis members are included in any design member. You may re-initialize any design member if you want to remove the linking.

The member design data must be edited for any member where the default member design data is not correct. Use the **Design > Design Data** command to do this. Default member design data, including the steel design code, is specified with the **File > Configure > Steel Design** command. The default member design data is displayed and may be altered for each group of members being initialized.

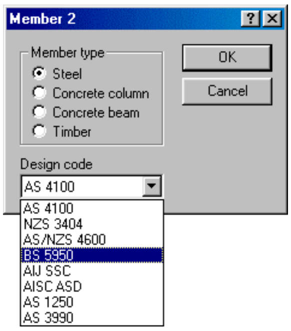
Design Data



DESIGN > DESIGN DATA

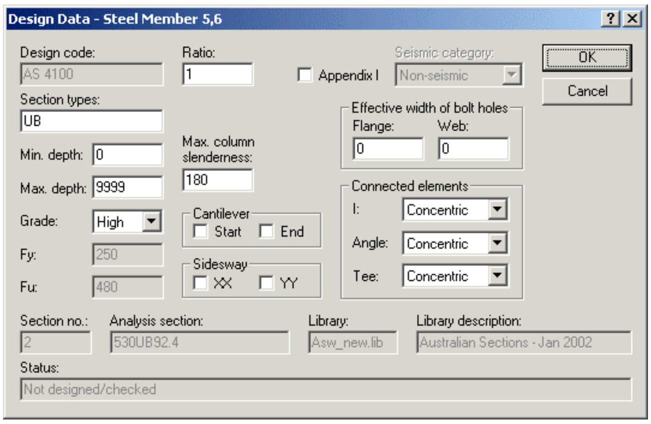
On selecting the **Design > Design Data > Input/Edit** command Microstran highlights in turn all the members so you can enter steel design data in a series of dialog boxes.

In the first dialog box, choose the design code from those available in the list box.



CHOOSING THE DESIGN CODE

The next step is to enter basic design data for the selected member in the dialog box shown below. You may click the ? button at the top of the dialog box for detailed information about each item.



DESIGN DATA

The dialog box contains the following items:

- **Select from these types**
  A list of the section types to be used in design. Section types are described by mnemonics (e.g. UB, UC). A space or comma should separate individual mnemonics. Examples of the selection list are:
  UB – Only UB sections will be considered.
  RHS SHS – The section may be either an RHS or an SHS.
  There are design rules for I and H sections (rolled and welded), channels, rectangular and square hollow sections, circular hollow sections, angles, tee sections, and compound sections. Each particular configuration of compound section to be considered (e.g. component designation, gap, and packer spacing) must be included in the steel section library. For compound sections, the inter-component shear is not checked.

- **Grade**
  Each library section has up to 3 standard grades, Normal, High, and extra. Select a grade from the list box. Only the grades actually available for the selected section types are shown. If you do not want one of the listed grades, select "Fy =" and enter the yield stress and the ultimate tensile stress in the edit boxes.

- **Depth range**
  Only sections with depths between the minimum and maximum values specified will be considered during design. The time required to search for a satisfactory section during design will be reduced if the minimum depth is increased to a realistic value.

- **Acceptance ratio**
  This is the limiting acceptance ratio – normally 1.0. To determine whether a member is adequate the *design ratio* (i.e. a load factor or a stress ratio, depending on the design code) is compared to this value. A member is adequate if its load factor *exceeds* this value or its stress ratio *is less than* this value.

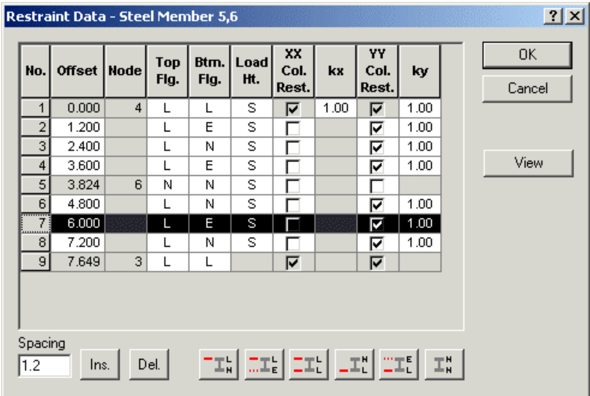
- **Cantilever**
  Check a box to indicate that either end of the design member is a cantilever. For linked design members the *start* is the exterior node of the first member in the linked list and the *end* is the exterior node of the last member in the linked list. For unlinked design members *start* is the "A" node and *end* is the "B" node.
- **Sidesway**
  Check a box to specify that sidesway is permitted about the respective section axis. The sidesway flags are used to determine the effective length for the computation of in-plane capacity. For AS 4100 and NZS 3404 they are also used in the computation of moment amplification factors where linear elastic analysis has been used.
- **Max. column slenderness**
  The maximum l/r ratio permitted. Typically 180 for columns and 300 for beams. This value is not used if not required by the design code in effect.
- **Effective width of bolt holes**
  The total effective width of any bolt holes in the flanges and webs of the members. The net area of the section will be obtained by deducting the cross-sectional area of the bolt holes. The width of bolt holes should be determined in accordance with the design code and should take into account the possibility of diagonal failure lines.
- **Connected elements**
  Specify whether the member is connected eccentrically with respect to the centroidal axis, giving rise to additional bending moments that must be considered in design (not applicable to hollow sections). For angles, you must specify whether the connection is concentric or through the long leg or the short leg. For an equal angle, the leg parallel to the section XX axis is considered to be the "short" leg. For channels and I sections, specify whether the section is connected concentrically or through the web or the flanges. For tee sections, specify whether the tee is connected concentrically or through the flange. Eccentric connection moments are considered for compression design only. The effects of eccentricity for tension members is taken into account by code rules that reduce the effective cross-sectional area.
- **Appendix I**
  For AS 4100 and NZS 3404 only. Specify whether webs will be checked using the combined stress provisions of Appendix I. The rules of Appendix I are not usually applied to standard sections used in building structures.

Key concept.

Lateral Restraint Data

Lateral restraints may be added at any point along a design member. At each restraint position you must specify the restraint at the top and bottom of the member, and any restraint provided for column action. The top of the member is defined as the +*y* face (*y* is the member axis usually aligned in the plane of the web of an I section – see “Coordinate Systems” on page 63). Use the **View > Display Options** command to show the *y* axis for all members. When the section XX axis is aligned with the member *y* axis, the top of the section is the +z face – see “Selection from Library” on page 89.

The dialog box shown below is used to specify the lateral restraint data for a particular design member.



No.	Offset	Node	Top Flg.	Btm. Flg.	Load Ht.	XX Col. Rest.	kx	YY Col. Rest.	ky
1	0.000	4	L	L	S	☑	1.00	☑	1.00
2	1.200		L	E	S	☐		☑	1.00
3	2.400		L	N	S	☐		☑	1.00
4	3.600		L	E	S	☐		☑	1.00
5	3.824	6	N	N	S	☐		☐	
6	4.800		L	N	S	☐		☑	1.00
7	6.000		L	E	S	☐		☑	1.00
8	7.200		L	N	S	☐		☑	1.00
9	7.649	3	L	L		☑		☑	

LATERAL RESTRAINT DATA

The items in the table are:

- **Offset**
  The distance from the end of the design member to the cross-section where the lateral restraint is being specified. Where there is a rigid offset at the end of the member, the offset distance is measured from the end of the rigid offset, not the node. You may insert a new restraint position by highlighting the row of the next offset and clicking the **Ins.** button. Delete an offset by highlighting it and clicking the **Del.** button. Any offset value may be changed as long as all offsets remain in ascending order.
- **Node**
  Where the position of the lateral restraint corresponds to a node the node number is shown in this column. You cannot change this value.
- **Top Flg. / Btm Flg.**
  Lateral torsional buckling (LTB) restraints are defined with the following codes:
  L – Effective lateral restraint.
  N – No restraint.
  C – Continuous lateral restraint.

R – Plan rotational restraint.
E – Elastic lateral restraint (provides twist restraint in combination with "L" restraint on opposing flange). This code was "P" in previous versions of Microstran.
Note: LTB restraints are ignored for sections not subject to lateral torsional buckling (i.e., CHS, SHS).

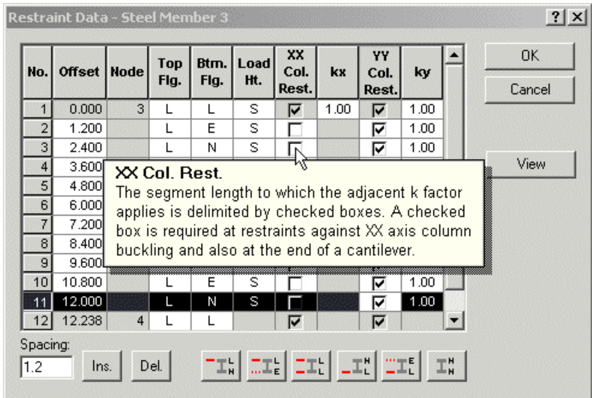
- **Load Height**
  Specify whether the load height position is such as to *Stabilize* or *Destabilize* the critical flange of the beam during beam bending. For end-restrained beams under gravity loads, loads applied to the top flange are destabilizing while those applied at the level of the shear centre or below are stabilizing. Loads applied at positions of restraint are always stabilizing. The effects of destabilizing loads are taken into account in most design codes by increasing the effective length for bending.

- **Column Buckling (In-Plane) Restraints**
  Specify the location of restraints against column buckling about the major and minor principal axes of the cross-section. The segment length to which the k factors apply is delimited by checked boxes. A checked box is required at restraints against column buckling and also at the end of a cantilever. Usually, any bracing that provides cross-sectional restraint will also provide restraint against buckling about the minor section axis but the converse may not be true. For example, a flexible bracing member connected to a beam at mid-height does not constitute a beam restraint that meets code requirements for cross-section restraint and so should be ignored when determining the effective segment length for out-of-plane buckling (LTB). Buckling restraint is specified independently for each restraint position.
  Note: For angle sections restraints are applied in principal axis directions.

- **kx and ky**
  kx and ky are the effective length factors for column buckling about the section XX and YY axes respectively. The member effective length is calculated as k × the distance between column buckling restraints. The effective length is required for the determination of nominal capacity for members in compression and, for some design codes, the computation of the moment amplification factor if linear elastic analysis has been used.
  Results from ECL Analysis
  If results are available from an elastic critical load (ECL) analysis, you may enter "ECL" in the kx and ky fields to have effective lengths computed from the elastic buckling load factors (λc). The results from an ECL analysis will only be applicable if the model includes all member restraints relevant to column action. *If you change column restraints during member design ECL analysis results may not be correct.* ECL analysis results will be most useful for structures modelled as plane frames where all out-of-plane

action is restrained. ECL analysis results must be available for all the load cases nominated for design. Effective length factors for out-of-plane column buckling cannot be computed if the structure has been modelled as a plane frame. See “Elastic Critical Load Analysis” on page 255.

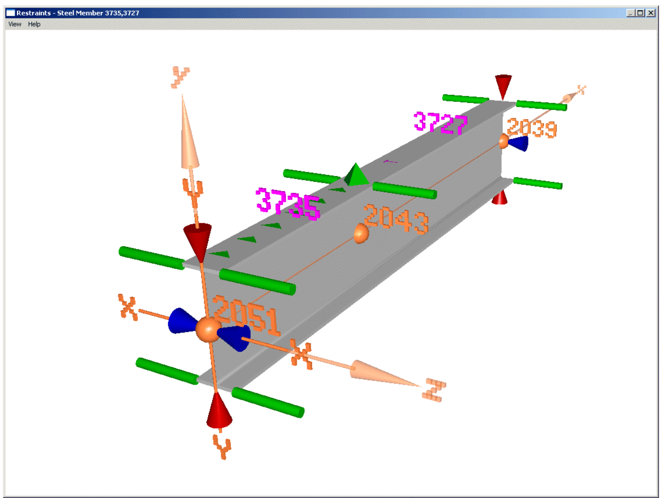
You may click on the ? button and then on any item in the dialog box for a pop-up help window.



POP-UP HELP

Graphical Display of Restraint Data

You may display a diagrammatic view (see below) of a steel design member by right-clicking on it and selecting the **View Steel Restraints** command on the context menu.



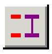
GRAPHICAL DISPLAY OF LATERAL RESTRAINTS

The design restraints are represented by the following symbols:

- Red cones – XX column buckling restraint.
- Blue cones – YY column buckling restraint.
- Green rods – effective lateral flange (LTB) restraint.
- Broken green rods – elastic flange (LTB) restraint.
- Green pyramid – rotational restraint of flange.
- Green triangles – restraints continuous over segment.

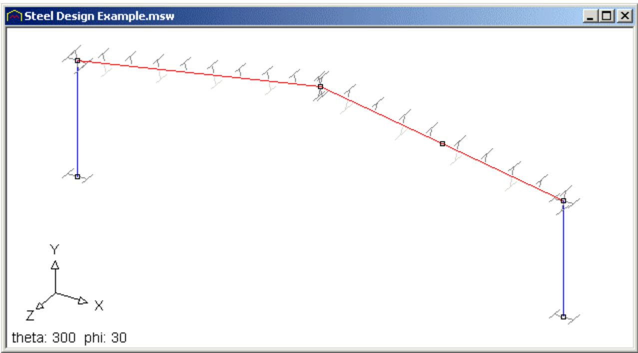
Nodes are shown as orange spheres and member numbers are shown in magenta above the top flange. The section axes XX and YY are shown, together with the member axes of the first link in the design member.

In classifying the restraint condition at a section, each physical restraint is interpreted according to whether the flange where the restraint occurs is critical.



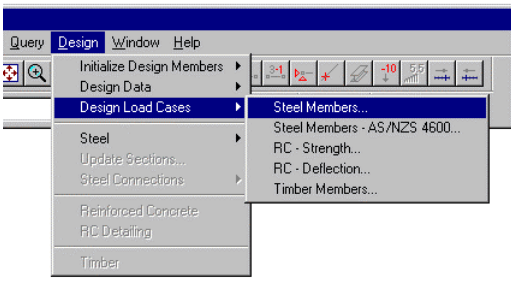
Steel Restraints

The Steel Restraints button on the Steel Design toolbar displays lateral torsional buckling restraints for all steel design members. Note that column buckling restraints are not shown. LTB restraints are shown as a short grey line at the flange positions of the member. The offset from the member is not to scale. “L” type LTB restraints are shown in a darker grey than those of type “E”.



DISPLAY OF LTB RESTRAINTS

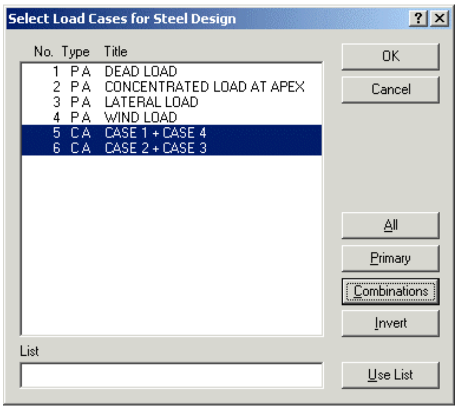
Selecting Design Load Cases



DESIGN > DESIGN LOAD CASES

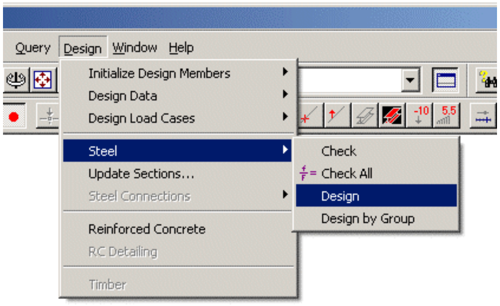
You must select the load cases to be considered in the steel design.

All load case titles are shown in a list box and you may select any of these by clicking on them. Sometimes, it might be more convenient to enter a list of load case numbers rather than selecting graphically. To do this, enter the list in the **List** edit box and click the **Use List** button. The rules for lists are set out in Chapter 15 – "Reports".



SELECTING LOAD CASES

The Design Process



DESIGN > STEEL

Check

The selected members are checked.

Check All

All steel design members are checked.

Design

A steel section is automatically chosen from the library for each of the selected members. The lightest suitable section is chosen independently for each design member. You may use this type of design as a basis for rationalizing section numbers throughout the structure in readiness for a final design using the Design by Group command.

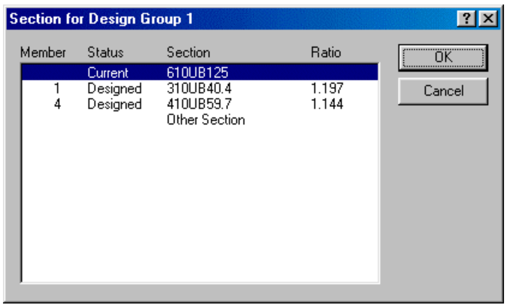
Design by Group

The section numbers define groups of members for which a single section will be chosen. Group design proceeds by alternate design and check cycles on successive members in the group until the lightest section that will be satisfactory for all members in the group has been chosen. Member groups may comprise single and linked members.

Updating Sections

For structures that are statically indeterminate, the internal forces in each member depend on the relative stiffness of the member. If the sections selected in the design are significantly different from those that were assumed for the analysis, you may update the input data by selecting the **Design > Update Sections** command. For each design group a dialog box (see below) displays all the different sections designed for members of the group. You may choose one of these sections, the original analysis section, or any other section from the library.

After re-analysis, you may check the design. On second and subsequent design cycles it is best to operate in *check* mode and to use *design* mode only for those members that fail the check.



UPDATING SECTIONS

Computations

Microstran computes member forces at a number of points along the member, including the ends of the member, points of application of concentrated loads, and all points of torsional restraint. Corrections are made to the analysis moments for eccentric connection and, for angles where rectangular axis properties were selected, the moments are resolved into the directions of the principal axes.

Note: Microstran Steel Design does not check torsion or bearing. Additional manual checks must be made for any member that is subject to significant torsion or high bearing forces.

Obtaining Design Results

After checking or designing members the design results may be displayed or reported in a number of ways:

- Use the **Results > Design Ratios** command to display design results with members colour-coded to show the percentage of member capacity actually utilized in the critical load case. With this display, all members that have failed a design check are shown in a shade of red. Click the Legend tab the output window to display the colour bands being used.
- Use the **Query > Design Member** command to show a summary of design results in the output window for any selected member.
- Use the **Reports > Design** command to create a design report. Members to be included in the report may be selected with the mouse or all members may be selected by checking the **All members** option button in the dialog box. The design report may be previewed with the **File > Print Preview** command and it may be printed with the **File > Print File** command. Note that there are extensive facilities for formatting the design report using the **File > Page Setup** command.

Steel design results are saved when you save a job but the steel design report file is automatically deleted (if it exists) when the job is closed.

The steel design report file is created in the data folder and it is named Job.p4, where “Job” is the job name. You may save a steel design report file by dragging it to another folder using Windows Explorer.

Steel Detailing

Information may be exported in SDNF format for transfer to third-party steel detailing programs (e.g. Xsteel, ProSteel). Member design must have been performed prior to selecting these output options.

17:Steel Connection Design

Overview

Steel connection design is an extension of the steel member design option that allows you to choose connection types for any design member and then check the connection. The steel member design and connection design options must both be available. Each connection type has default data to minimize input of design data. The connection dialog box may be displayed by right-clicking on the design member at the connection location.

Any design member may have a connection at each end and a single splice at a specified distance from the start of the design member. In some cases you have a choice of which design member you allocate the connection to – for example, an apex connection in a portal frame may be associated with either rafter. It does not matter which member the connection is associated with but you should not input the same connection twice.

Microstran's virtual reality window shows the structure with the steel connections. The connection dialog box can also be displayed by right-clicking on a connection in this view.

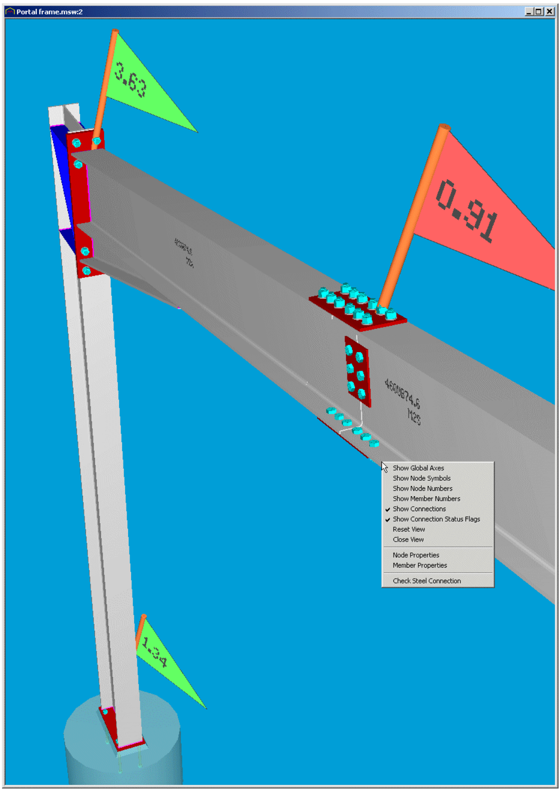
The following restrictions apply to Microstran models containing connections:

- Any member with a connection must have the member *y* axis parallel to the section YY axis (the web plane).
- Any truss with connections must be in a vertical plane.
- Only two branches are permitted in the KT gusset plate connection (KTG).
- The beam/column diagonal bracing connection (UFBR) is not yet available.

Virtual Reality Graphics in Microstran

An important feature of Microstran is its virtual reality (VR) view. When the connection design option is available this graphical representation may include every detail of each connection in the model, right down to the welds and markings on the bolts. Each connection is shown in the connection dialog box in a standard position. When the same connection is shown in Microstran it must be moved into the position where it exactly fits the rest of the structure. This usually means that the connection appears in Microstran with a different orientation from that shown in the connection dialog box.

Initially, when you drag the mouse, the structure rotates in the drag direction about the centre of the structure. Double-click on any point on the displayed model and that point becomes the new centre of rotation. If the double-click location is not on the structure the structure may disappear – in this case, press **Esc** to revert to the initial view. You may pan the view by dragging with the right mouse button and you may zoom with the mouse wheel or the **Page Up** and **Page Down** keys.



CONNECTION IN MICROSTRAN'S VIRTUAL REALITY VIEW

A pop-up menu appears when you right-click the stationary mouse anywhere on the structure (if the mouse is moving you will pan). This menu lists keyboard shortcuts to toggle display items.

C – Connections

X – Connection symbols

F – Connection status flags

N – Node numbers

S – Node symbols

M – Member numbers

It also allows you to display node or member property dialog boxes and change or check a connection.

Connection status flags show the connection number on one side and the connection strength ratio (load factor) on the other – type **R** to reverse the flags. The flag is green when the strength ratio is 1.0 or greater, red when it is less than 1.0, and black when the connection has not been checked.

The **File** menu for the VR view allows you to save a .JPG file of the image or print the image. Keyboard shortcuts for these commands are **J** and **P**, respectively.

Simple Shear Connections

Three simple shear connections are available. These are:

- WSP – Web side plate (also known as a fin plate or shear tab)
- ACLT – Angle cleat
- FEP – Flexible end plate

These connections can be selected at either end of any horizontal I section or channel design member. Microstran identifies the support member, which may be a column or a beam. The connection design ignores any bending moment in the Microstran model. At locations where you specify shear connections you should ensure that the Microstran model has a moment release or negligible bending moment.

Moment Connections

Moment connections available are:

- BMEP – ASI* bolted moment end plate
- HBEP – ASI* haunched end plate
- WBC – ASI* welded beam/column
- MEPC – AISC† moment end plate connection
- FPC – AISC† flange plate connection (welded or bolted)

  * Australian Steel Institute model
  † American Institute of Steel Construction model

These connections can be selected at either end of non-vertical I section design members. The MEPC and FPC connections can only be in horizontal members. Microstran identifies the support member, which must be a column oriented so that the connection is to the column flange.

Only the connection between the beam and the column is checked. Column actions and actions from other beams connected to the column are ignored.

Column stiffeners may be specified for any of these connections. They are checked ignoring the effect of member forces in the column and any other beam connected to the column.

Location of Haunched Connection

Haunched rafters may be modelled as a prismatic rafter and one or more larger sections representing the haunch. The prismatic part of the rafter would be a design member but the haunch members may not be. This can cause difficulty because connections cannot be input for members that are not design members. To input the haunched connection in this situation you must select a *remote* HBEP connection at the end of the prismatic rafter.

Splices

Splices available are:

- BMEP – ASI bolted moment end plate
- BSPL – ASI bolted splice
- WSPL – ASI welded splice

A single splice may be included for any I section design member. The distance from the start of the design member must be specified. The splice location does not have to coincide with a node.

Bracing Connections

Microstran permits several types of single bracing connection (BRAC). These allow you to check the connection between a bracing member and a cleat or gusset plate. Only axial forces in the bracing member are considered.

Chevron brace connections can be checked using the KT gusset plate connection (KTG). The beam/column diagonal bracing connection (UFBR) is not yet available in Microstran.

Base Plates

Pinned and moment base plates are available. Base plates are permitted only at vertical column supports. You should not specify a moment base plate at a pinned support or a pinned base plate at a support where there is bending moment.

HSS Truss Connections

General

HSS truss connections are available for trusses located in a vertical plane. Every member in the truss must have its member *y* axis in the plane of the truss and must be defined as a steel design member.

The types of HSS truss connection available are:

- K/N overlap or gap connection
- X connection
- Y/T connections

K/N and X connections have two bracing members, referred to in the connection dialog box as brace 1 and brace 2, while the Y/T connection has only one.

Each connection is associated with one of the design members at the joint. *Where chord members have been initialized linked it is necesssary to associate the connections with bracing members.* .This is because the members in each chord form a single design member that can have connections only at its ends.

For the X connection, the two members with the greater section depth are assumed to be the chord members. If the chord and bracing sections are of the same depth the member associated with the connection is assumed to be a chord member.

For HSS truss connections Microstran evaluates all relevant member forces at the connection and transfers to the connection dialog box all resultants necessary for checking the connection according to the connection design model.

Limitations of the K/N Connection Model

The models for the K/N gap and overlap connections are only valid for one bracing member in tension and the other in compression. A node load occurring at the connection may result in both bracing members being simultaneously in tension or compression, thus invalidating the model. If Microstran detects this condition for any load case an error message is displayed.

Noding Eccentricity

The eccentricity at a connection is the distance from the intersection of the bracing members to the centre-line of the chord. Eccentricity is positive when measured towards the outside of the chord and negative towards the inside.

Initially, all K/N truss connections are shown in the connection dialog box with a zero chord eccentricity. Whether K/N connections with zero eccentricity are gap or overlap connections depends on the geometry of

the truss. This means that a connection specified in Microstran as a K/N overlap connection may appear in the connection dialog box as a K/N gap connection. Similarly, a connection specified in Microstran as a gap connection may appear in the connection dialog box as an overlap connection. When you edit a connection you may set the eccentricity as required and the connection type will be adjusted accordingly.

The values of gap or overlap corresponding to zero eccentricity may not be admissible in the connection design model – for example, in an overlap connection the overlap must be at least 25% but cannot exceed 100%. Usually, the gap or overlap would be specified to give the desired connection detail but the resulting eccentricity must then be considered.

Unless rigid offsets are specified in the chord members the eccentricity in the Microstran model will be zero. Discontinuities will be visible in the Microstran VR view between a connection and the rest of the structure if the eccentricity in the Microstran model does not equal that in the connection.

Joint Bending Moments

Bracing Members

Bending moment is usually not significant in the bracing members of a truss, and indeed, it is not considered in the HSS truss connection models. The bracing members in the Microstran model for an HSS truss often include pins (moment releases) at each end but bracing member bending moments are usually negligible without them.

Chords

The HSS truss connection models take into account bending moment in the chords caused by transverse member loading but that caused by noding eccentricity may be ignored provided that the eccentricity is within the prescribed limits. For design of the chord *members*, however, the bending moment caused by noding eccentricity must be considered. It is necessary to introduce rigid offsets in the chord members to ensure that the chord design moments are consistent with the connection eccentricities.

Using Steel Connection Design

Note: You must use kN and meter units in your Microstran model if you are going to use steel connection design.

Steel Member Design

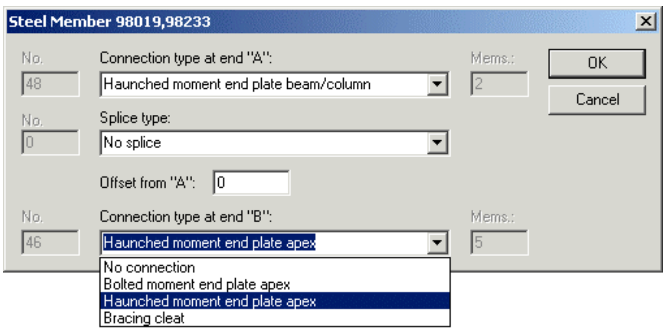
Before using integrated connection design you must have used an optional steel design module to check all design members. The steps required are:

- Analyse the structure.
- Initialize design members.
- Modify design data as required.
- Specify the load cases to be used in the design.
- Check all design members.
- Update design member sections as required.
- Final check of all design members.

Please refer to Chapter 16 – "Steel Member Design" for detailed information.

Input Connection Data

The only item of connection data required in Microstran is the connection type, and for a splice, the offset from the start of the member. The dialog box below is displayed for each design member where you wish to add a connection. Not all connection types are displayed in the drop-down list boxes because any types that are not feasible are automatically suppressed. At the bottom of a column, for example, the only available connection types are the pinned and fixed base plate. The connection number allocated by Microstran is shown to the left of the box showing the connection type. On the right of this box the number of connected members, not including the connection member, is shown.



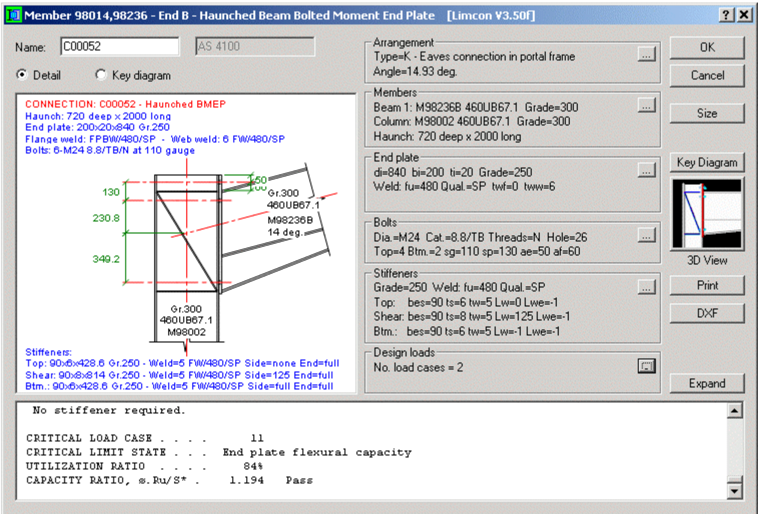
CONNECTION TYPE DIALOG BOX

This dialog box is displayed after the steel member design data dialog boxes when you select the **Design > Design Data > Input/Edit** command. You may also go straight to this dialog box by right-clicking any design member and choosing **Steel Connection Type** on the pop-up menu.

If an inappropriate connection has been chosen an error message will normally be displayed when you check the connection.

Check Steel Connections

The next step in connection design is to select the **Design > Steel Connections** command and then click on all design members whose connections are to be checked. After selecting these design members, right-click and choose **OK** on the pop-up menu. A connection dialog box is then displayed for each connection in turn.



CONNECTION DIALOG BOX

You may change any connection detail or dimension. The dialog box for some simpler connections contains a **Design** button, which iteratively changes the connection parameters, searching for a configuration that passes all strength checks. This may not always give a satisfactory design. It is your responsibility to ensure that the final connection configuration is satisfactory in every way.

When you click the **OK** button in the connection dialog box the connection details are saved and the dialog box is displayed for the next connection. If you click the **Cancel** button any changes made in the dialog box are abandoned and the dialog box is not displayed for any remaining selected connections.

You may check connections one at a time by right-clicking on a design member and selecting **Check Steel Connection** from the pop-up menu. Where you right-click on the design member determines which

connection is checked. Each end quarter of the design member selects the connection at the end if it exists and the middle half of the design member selects the splice if it exists.

Display Steel Connections

Having checked all the connections in the job, you may display a VR image of the whole structure including connections by clicking the **Virtual Reality View** button. In the initial view connections are not shown – type **C** to redraw the view including connections.

Report

A detailed report for selected connections is available with the **Reports > Steel Connections** command. The report may be previewed with the **File > Print Preview** command or printed using the **File > Print File** command.

Interaction with Limcon

If the Limcon program is available you can see all the connections for a Microstran job in a single view by starting Limcon with this command line:

```
Lmc3.exe /ALL=jobname
```

Where "jobname" is the Microstran job name.

The shortcut must start Limcon in the Microstran data folder. Limcon collects the Microstran connection files for the job and shows all the connections together. Any changes you make to the connections at this stage will be inaccessible to Microstran.

Connection Design Example

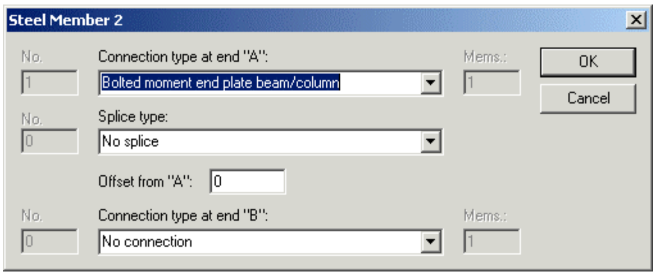
The following step-by-step example illustrates the design of a knee connection in the portal frame from Example 2 in Chapter 20.

Step 1 – Check steel member design results

The status of the steel member design can be checked by clicking the **Design Ratios** button on the Results toolbar and the **Display Results Values** button on the Results toolbar. This shows design members colour-coded according to the design ratio.

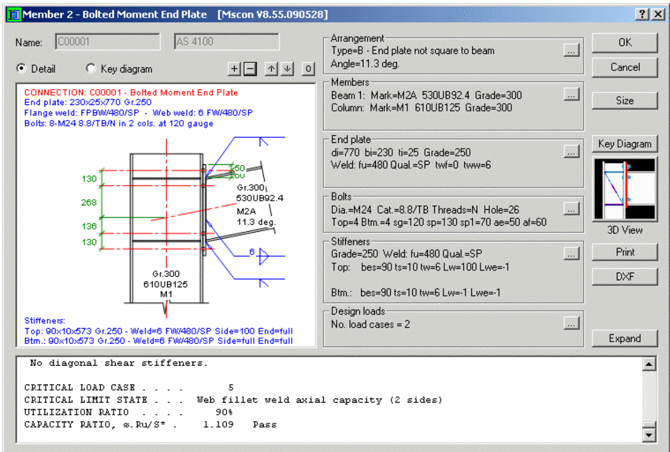
Step 2 – Choose connection type

Right-click on rafter near the knee joint and select **Steel Connection Type**. Select the bolted moment end plate connection type.



Step 3 – Initial check of steel connection

Right-click on the rafter again and select **Check Steel Connection**. The connection design dialog box displays the initial configuration. The end plate has been sized to match the rafter and column members; other data items remain at default values.

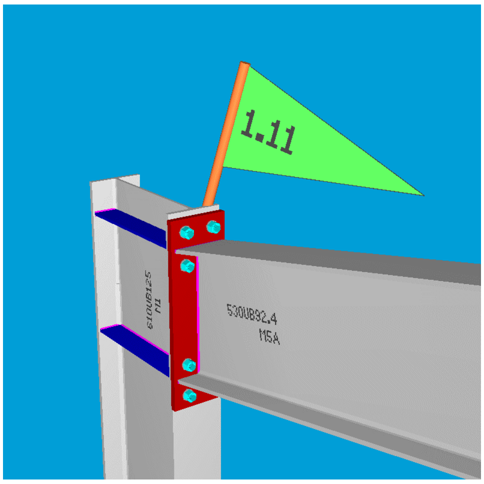


Step 3 – Modify connection as required

Click the Expand button to inspect the connection check output. Notice in Check 1 that the bolt gauge is less than the recommended value of 140 and in Check 22 that the stiffener width is less than the recommended minimum of 99. Change these values and then click **OK**.

Step 4 – Display VR image for structure with connections

Click the **OK** button to dismiss the connection dialog box and in Microstran click the **Virtual Reality View** button. Type **F** to display connection flags and **R** to reverse the flags showing the connection load factor. The resulting view is shown below.

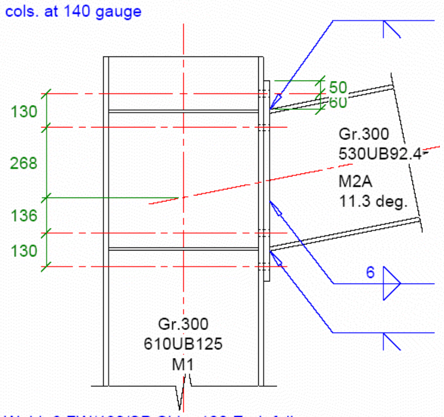


Step 5 – Print output for connection

Click the **Print** button to output the design check for the connection. This output is shown on the following pages.

Engineering Systems Development Page 1 of 4
Job: Ex2_c00001.0l1 -- Connection: C00001 28-May-09
01:11:04 PM

CONNECTION: C00001 - Bolted Moment End Plate
End plate: 230x25x770 Gr.250
Flange weld: FPBW/480/SP - Web weld: 6 FW/480/SP
Bolts: 8-M24 8.8/TB/N in 2 cols. at 140 gauge



Stiffeners:
Top: 100x10x573 Gr.250 - Weld=6 FW/480/SP Side=100 End=full
Btm.: 100x10x573 Gr.250 - Weld=6 FW/480/SP Side=full End=full

```
Geometry Check:
*** WARNING -- Check outer flange face distance.
*** WARNING -- Check inner flange face distance.
 2 warning(s)

MSCON V8.6                                                   28-MAY-09
                                                              13:06:12
 Connection: C00001
       Type: Bolted Moment End Plate Connection
             B - End plate not square to beam
    Country: Australia
      Units: SI metric
Design code: AS 4100

Beam 1:  Mark=M2A  Section=530UB92.4   Grade=300   Angle= 11.3 deg.
     D =    533 mm   Root rad. =     14 mm    fyf =    300 MPa
     B =    209 mm        Area = 1.1800E+04   fyw =    320 MPa
    Tf =     16 mm          Zx = 2.0800E+06    fu =    440 MPa
    Tw =     10 mm          Sx = 2.3700E+06
 Section bending capacity  . . . .   639.9 kN.m
 Section shear capacity  . . . . .   939.4 kN
 Section axial capacity  . . . . .  3281.0 kN

Column:  Mark=M1  Section=610UB125   Grade=300
     D =    612 mm   Root rad. =     14 mm    fyf =    280 MPa
     B =    229 mm        Area = 1.6000E+04   fyw =    300 MPa
    Tf =     20 mm          Zx = 3.2300E+06    fu =    440 MPa
    Tw =     12 mm          Sx = 3.6800E+06
 Section bending capacity  . . . .   927.4 kN.m
 Section shear capacity  . . . . .  1179.8 kN
 Section axial capacity  . . . . .  4158.4 kN
 Column terminates...
  Top flange to end of column  . .     200 mm

End plate:
 770x230x25  Gr./fy/fu=250/250/410 MPa
 Beam to end plate angle . . . . .   78.69°

Welds:
 FPBW fu=480 MPa/SP flanges.
```

Mscon V8.55.090528 [888888] D:\Mswin\Examples\Ex2_c00001.0l1

Engineering Systems Development
Job: Ex2_c00001.0l1 -- Connection: C00001

Page 2 of 4
28-May-09
01:11:04 PM

```
 6 FW fu=480 MPa/SP web.

Bolts:
 4-M24 8.8/TB/N top flange, 140 gauge.
 4-M24 8.8/TB/N btm. flange, 140 gauge.

Stiffeners:  Gr./fy/fu=250/260/410 MPa   Welds fu=480 MPa/SP
 2/100 x10 top, 6 FW 100 at midpoint and across ends.
 2/100 x10 btm., 6 FW full length and across ends.

MINIMUM ACTION CHECK
Specified minimum design actions:
 Bending       30% of øMs (  639.9)  =  192.0 kN.m
 Shear          0% of øVs (  939.4)  =    0.0 kN
                                         40.0 kN
 Tension        0% of øNs ( 3281.0)  =    0.0 kN
 Compression    0% of øNs ( 3281.0)  =    0.0 kN
 NOTE: Input design actions are not automatically increased if they are less than
       the specified minimum actions. Minimum actions may be set in any load case.
       This check warns if any design action is less than the specified minimum
       for all load cases.
 WARNING: Design bending moment is less than specified minimum.

DESIGN CHECK SUMMARY
  No.  Case       M1      V1      N1      Vc      M2      V2      N2      LF   Util.
   1     5     119.2     -55     -44       0     0.0       0       0    1.11    90%  <-- Critical
   2     6     -28.6      -6     -46       0     0.0       0       0    1.11    90%

INPUT DESIGN ACTIONS FOR CASE NO.    5
 Beam 1:  Moment, M* . . . . . . .   119.2 kN.m
          Shear, V*  . . . . . . .   -54.6 kN
          Axial, N*  . . . . . . .   -44.2 kN  (comp.)
 Column:  Shear, V*c . . . . . . .     0.0 kN
          Compression, N*c . . . .     0.0 kN

SECTION ANALYSIS RESULT
Simplified analysis:
 Beam 1...             N*ft =   198.8t   N*fc =   252.8c
                       N*wt =     0.0    N*wc =     0.0
                       M*w  =    0.00
                       V*w  =    49.7
Elastic analysis:
 Beam 1...             N*ft =   169.8t   N*fc =   194.3c
                       N*wt =     0.0    N*wc =    29.5c
                       M*w  =    23.1
                       V*w  =    49.7
Plastic analysis:
 Beam 1...             N*ft =   165.7t   N*fc =   190.0c
                       N*wt =     0.0    N*wc =    29.7c
                       M*w  =    25.3
                       V*w  =    49.7

Using ASI 2009 model...

NOTE: Simplified analysis results used.

BOLT ARRANGEMENT
 4-Bolt Extended End Plate - Unstiffened
 Connection checked for tension at top flange.

GEOMETRY CHECKS...
CHECK 1 - Detailing Recommendations:
 Ref. 43: Design Guide 12 - Bolted End Plate to Column Moment Connections
          Australian Steel Institute - 2009
 Bolt diameter . . . . . . . . . .      24          ≥      20                Pass
                                        24          ≤      24                Pass
 Bolt UTS  . . . . . . . . . . . .     830          ≥     827                Pass
 End plate thickness, ti . . . . .      25          ≥      16                Pass
                                        25          ≤      40                Pass
 End plate width, bi . . . . . . .     230          ≥     229                Pass
                                       230          ≤     249                Pass
 Bolt gauge, sg  . . . . . . . . .     140          ≤     171                Pass
                                       140          ≤     180                Pass
                                       140          ≥     140                Pass
 Bolt spacing, sp  . . . . . . . .     130          ≥      80                Pass
 Bolt edge distance, ae  . . . . .      50          ≥      36                Pass
                                        50          ≤      60                Pass
 Bolt flange face distance, af . .      60          ≥      40                Pass
                                        60          ≤      75                Pass
 NOTE: Clearances should be checked in virtual reality view.
```

Mscon V8.55.090528 [888888]   D:\Mswin\Examples\Ex2_c00001.0l1

Engineering Systems Development
Job: Ex2_c00001.0l1 -- Connection: C00001

Page 3 of 4
28-May-09
01:11:04 PM

```
DESIGN CAPACITY CHECKS...
                                                    Capacity ratio ──────────┐
                                                    Design action ──┐        │
                                                 ┌─ Design capacity │        │
                                                 │                  │        │
Section Bending/Axial:
 Flange tension yield capacity . . . . . . .   880.3  ≥  N*ft  =  165.7  5.31  Pass  Manual p.54
 Flange tension rupture capacity . . . . . .  1097.5  ≥  N*ft  =  165.7  6.62  Pass
 Flange compression capacity . . . . . . . .   880.3  ≥  N*fc  =  190.0  4.63  Pass

CHECK 2 - Flange Welds:
 Check not required for FPBW.

CHECK 3 - Web Welds:
 Web shear force . . . . . . . . .    49.7 kN
 Web bending moment  . . . . . . .    0.00 kN.m
 Length of web weld  . . . . . . .     484 mm
 Length for shear resistance . . .     242 mm
 Using 6 fillet weld, fu= 480/SP, both sides...
 Web fillet weld shear capacity (2 sides)  .   472.9  ≥  V*    =   49.7  9.51  Pass
 Section web axial capacity  . . .   2.938 kN/mm
 Minimum weld design force . . . .   1.763 kN/mm
 Section moment/capacity ratio . .   0.186
 » Web weld design force . . . . .   1.763 kN/mm
 Web fillet weld axial capacity (2 sides)  .   1.955  ≥  n*w   =  1.763  1.11  Pass
 NOTE: This check uses method from AISC SDG 13 (Ref. 20).

CHECK 4 - Bolts at Tension Flange:
 Single bolt tension capacity  . .   234.4 kN
 Sum of bolt lever arms  . . . . .    1061 mm
 End plate design moment, M*eq . .   119.2 kN.m
 Tension bolt moment capacity, øMbt  . . . .   497.5  ≥  M*eq  =  119.2  4.17  Pass

CHECK 5 - Bolts in Shear:
 Single bolt shear capacity  . . .   133.4 kN
 No. bolts in compression region .       4
 Bolt shear capacity, øVfb . . . . . . . . .   533.5  ≥  V*    =   49.7  10.7  Pass  ASI DG12 p.33
 End plate bolt bearing capacity, øVbi . . .  2833.9  ≥  V*    =   49.7  57.0  Pass  ASI DG12 p.33

CHECK 6 - End Plate in Bending:
 End plate design moment, M*eq . .   119.2 kN.m
 Yield line parameter, Yp  . . . .    3624 mm
 End plate moment capacity, øMpt . . . . . .   509.6  ≥  M*eq  =  119.2  4.28  Pass
 Bolt moment capacity, øMbt  . . .   497.5 kN.m
 Section moment capacity, øMs  . .   639.9 kN.m
 » No prying capacity, Mmin  . . .   552.2 kN.m
 » End plate no prying thickness .      27 mm                                    SDG4 3.10
 » Thin plate behaviour governs...
 Single bolt tension capacity  . .   234.4 kN
 Specified minimum tension . . . .   210.0 kN
 Bolt max. prying force, Qmax,i  .    30.4 kN
 Bolt max. prying force, Qmax,o  .    64.1 kN
 Bolt rupture capacity (prying)  . . . . . .   413.7  ≥  M*eq  =  119.2  3.47  Pass
 Prying factor . . . . . . . . . .    0.38                                       Informative
 Bolt efficiency . . . . . . . . .     83%                                       Informative

CHECK 7 - End Plate in Shear:
 Horiz. shear  . . . . . . . . . .    99.4 kN
 Horiz. shear yielding capacity, øVpe  . . .   646.9  ≥  V*h   =   99.4  6.51  Pass  ASI DG12 p.38
 Horiz. shear rupture capacity, øVpu . . . .   985.2  ≥  V*h   =   99.4  9.91  Pass  ASI DG12 p.38

CHECK 8 - End Plate Stiffener:
 No stiffener.

CHECK 9 - End Plate Stiffener Welds:
 No stiffener.

COLUMN-SIDE CHECKS

CHECK 10/16 - Unstiffened Column Flange Bending at Beam Tension Flange:
 NOTE: This capacity is required for checking stiffeners.
 Capacity reduced if column terminates within 150mm of top bolt row.
 Top bolt row to end of column . .      140          ≥    150               Fail
 » Capacity reduced for terminating column.
 Yield line parameter, Yc  . . . .    3830 mm
 Col. flange no prying thickness .      25 mm                                    SDG4 3.20
 Column flange capacity  . . . . .   370.8 kN.m
 Equivalent flange force, øRft . .   702.7 kN                                    ASI DG12 p.61
 End plate design moment, M*eq . .   119.2 kN.m
```

Mscon V8.55.090528 [888888] D:\Mswin\Examples\Ex2_c00001.0l1

Engineering Systems Development Page 4 of 4
Job: Ex2_c00001.0l1 -- Connection: C00001 28-May-09
01:11:04 PM

```
 No prying capacity, Mmin  . . . .   552.2 kN.m
 Unstiffened col. flange capacity, øMct  . .   370.8  ≥  M*eq  =  119.2  3.11  Pass  Informative
 » Tension flange stiffeners may not be required.

CHECK 11 - Unstiffened Column Web Yielding at Beam Tension Flange:
 NOTE: This capacity is required for checking stiffeners.
 Capacity reduced if column terminates within Dc of top flange.
 Top flange to end of column . . .      200          ≥     612               Fail
 » Capacity reduced for terminating column.
 Unstiffened col. web yield capacity, øRwt .   454.3  ≥  N*ft  =  198.8  2.28  Pass  Informative
 » Tension flange stiffeners may not be required.

CHECK 12 - Unstiffened Column Web Yielding at Beam Compression Flange:
 NOTE: This capacity is required for checking stiffeners.
 Unstiffened col. web yield capacity, øRwy .   958.3  ≥  N*fc  =  252.8  3.79  Pass  Informative
 » Compression flange stiffeners may not be required.

CHECK 13 - Unstiffened Column Web Crippling at Beam Compression Flange:
 NOTE: This check not required with compression flange stiffeners.
 Column web crippling capacity, øRwc . . . .   875.2  ≥  N*fc  =  252.8  3.46  Pass  Informative
 » Compression flange stiffeners may not be required.

CHECK 14 - Unstiffened Column Web Buckling at Beam Compression Flange:
 NOTE: This check not required with compression flange stiffeners.
 Column web buckling capacity, øRwc  . . . .   936.5  ≥  N*fc  =  252.8  3.70  Pass  Informative
 » Compression flange stiffeners may not be required.

CHECK 15/21 - Unstiffened Column Web Panel in Shear:
 Column web panel shear  . . . . .   198.8 kN
 Column web panel shear capacity, øVp  . . .  1179.8  ≥  V*p   =  198.8  5.93  Pass  ASI DG12 p.50

CHECK 22 - Transverse Stiffeners at Beam Tension Flange:
 Stiffener width . . . . . . . . .      100          ≥      99               Pass
                                        100          ≤     109               Pass
 Stiffener effective width . . . .     100 mm
 Stiffener thickness . . . . . . .       10          ≥       8               Pass
 Column flange capacity, øRft  . .   702.7 kN
 Column web yield capacity, øRwt .   454.3 kN
 » Unstiffened column capacity . .   454.3 kN
 Flange tension  . . . . . . . . .   198.8 kN
 » Check of stiffeners not required.

CHECK 22A - Stiffened Column Flange Bending at Beam Tension Flange:
 Yield line parameter, Ycs . . . .    5703 mm
 Col. flange no prying thickness .      21 mm                                        SDG4 3.20
 Column flange capacity  . . . . .   552.1 kN.m
 End plate design moment, M*eq . .   119.2 kN.m
 Stiffened col. flange capacity, øMcts . . .   552.1  ≥  M*eq  =  119.2  4.63  Pass
 No prying capacity, Mmin  . . . .   552.2 kN.m
 » Bolt prying does not exist.

CHECK 23 - Transverse Stiffeners at Beam Compression Flange:
 Stiffener width . . . . . . . . .      100          ≥      99               Pass
                                        100          ≤     109               Pass
 Stiffener thickness . . . . . . .       10          ≥       8               Pass
 Stiffener side weld . . . . . . .      543          ≥     523               Pass
 Design compression, N*cs  . . . .   252.8 kN
 Column web yield capacity, øRwy .   958.3 kN
 Stiffener section yield...
  Stiffener yield capacity . . . .   468.0 kN
 Stiffener/web cruciform section yield...
  Yield capacity, øRfcy  . . . . . . . . . .  1426.3  ≥  N*cs  =  252.8  5.64  Pass
 Stiffener/web cruciform section buckling...
  Buckling capacity, øRfcb . . . . . . . . .  1689.6  ≥  N*cs  =  252.8  6.68  Pass
 Side welds checked for force in excess of unstiffened column web capacity...
 » Weld design force . . . . . . .     0.0 kN
  Total side weld length . . . . .    2171 mm
  Stiffener side weld capacity, øRfcw  . . .  2122.4  ≥  N*cs  =    0.0  0.00  Pass

CHECK 24 - Diagonal Shear Stiffeners:
 No diagonal shear stiffeners.

CRITICAL LOAD CASE . . . .        5
CRITICAL LIMIT STATE . . .  Web fillet weld axial capacity (2 sides)
UTILIZATION RATIO  . . . .      90%
CAPACITY RATIO, ø.Ru/S* .    1.109   Pass
```

Mscon V8.55.090528 [888888] D:\Mswin\Examples\Ex2_c00001.0l1

18:Section & Material Libraries

General

Microstran refers to the current steel section library for information required for analysis, checking, and design of steel members. The **File > Configure > Section Library** command allows you to select any available library as the current library. Using the method described below you may edit the library. Special section libraries may also be created.

Section Library

Microstran's library files have the file name extension "lib" (e.g. Asw.lib, Ukw.lib) and cannot be listed, printed, or edited. For each library file there is a corresponding source file, an ordinary text file having a file name extension "asc". Library source files may be manipulated by the Section Library Manager. The library name must not exceed 8 characters in length and it must not be a number.

Section Name

Each section has a unique section name with up to 15 characters. Blanks are not permitted. The section name must have one contiguous alphabetic group between 1 and 4 characters long. This is the *section mnemonic*.

Section Mnemonic

The section mnemonic is used in Microstran for specifying sections to be chosen automatically in design. It is embedded in the section name and, apart from "X", is the only part of the name that may be alphabetic. An "X" character contiguous with the section mnemonic is part of the section mnemonic. Apart from the section mnemonic, "X" characters with numeric characters before and after may be included in the section name.

Examples of valid section names are, "200UB25.4", "88.9X2.6CHS", "CTT380X100", "100XX", "XX100", and "W14x311". Invalid names include "200UB25.4H1" (two separate alphabetic groups),

“CTT380X100X” (trailing X), “X200UB25.4” (leading X), and “XXBOX100” (mnemonic exceeds 4 characters).

When adding new sections to a library you may choose any suitable section mnemonic. A single character “E”, however, may not be used as a section mnemonic because the section name would then be confused as a number in exponential format.

Section Categories

Each group of sections in the library is assigned a *section category*. Every section in a section category must have the same section mnemonic. The section category number is shown in the library source file under the heading SC. When choosing a section, you first choose the section category and Microstran then displays all the sections in the category. All sections within a category must have the same design type and section mnemonic.

Design Type

For design purposes each section is classified according to its *design type*. The design type number is shown in the library source file under the heading DT. The design type is used to interpret the section properties and it determines the applicable design code rules. The table below lists valid design types, together with some of the common section mnemonic codes for these types.

DT	Mnemonic	Section Type
1	TFB	Taper flange beam
2	UB, WB	Universal beam or welded beam
3	UC, WC	Universal column or welded column
4	RHS	Rectangular hollow section
5	SHS	Square hollow section
6	CHS	Circular hollow section
7	PFC	Parallel flange channel
8	BT, CT	Tee section
9	EA	Equal angle
10	UA	Unequal angle
11	DAL	Double angles, long legs together
12	DAS	Double angles, short legs together
16	STA	Starred angles
22	QAN	Quad angles
13	UBP	Universal bearing pile
17	TFC	Taper flange channel
18	ROD	Round
19	BAR	Rectangular bar
20	CTT	Double channels, toes together

21	CBB	Double channels, back-to-back
24	CA	DuraGal cold-formed angle
25	CC	DuraGal cold-formed channel
30	-	Section with analysis properties only
33	UI	Unsymmetrical I section
34	BOX	Box section
39	LSB	OneSteel LiteSteel™ Beam
40	DLSB	Double OneSteel LiteSteel™ Beam
41	PEA	Plain cold-formed equal angle
42	PUA	Plain cold-formed unequal angle
43	PC	Plain cold-formed channel
44	DPC	Double plain cold-formed channel
45	LC	Cold-formed lipped cee
46	DLC	Double cold-formed lipped cee
47	RLC	Cold-formed return lipped cee
48	DRLC	Double cold-formed return lipped cee
49	PZ	Cold-formed plain zed
50	LZ	Cold-formed return lipped zed
51	DHS	Dimond Hi-Span™ section
52	DDHS	Double Dimond Hi-Span™ section

Steel Grades

Microstran determines yield and ultimate tensile stresses from recognized steel grades. Up to three grades, G1, G2, and G3, may be included for each section. If there is only one grade available for a section, enter it as G1 and enter zero for G2 and G3. If there are two grades available, G3 must be zero.

Residual Stress Code

Some design codes (e.g. AS 4100) require information about the level of residual stresses in a section. This is provided by the parameter designated “f”.

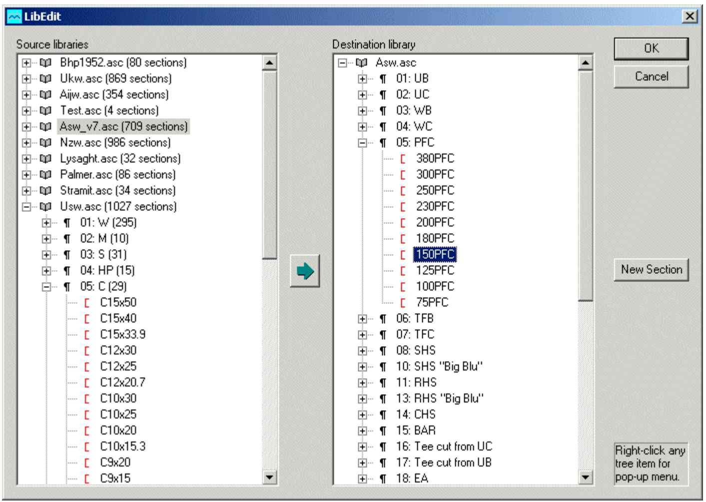
f	Section Type
1	Stress relieved
2	Hot-rolled
3	Cold-formed
4	Lightly welded
5	Heavily welded

Section Library Manager

You may edit any library source file supplied but it is preferable to make a copy and edit that – otherwise, you will lose your changes when you next update library files.

Library source files may be manipulated by the Section Library Manager.

After you have selected the destination library, either an existing library source file or a new one the dialog box below is displayed. A tree view of the destination library, empty if new, is shown on the right while all available library source files are shown on the left. Each library may be expanded to show the section categories and each of these may be expanded to show the sections contained in the category.

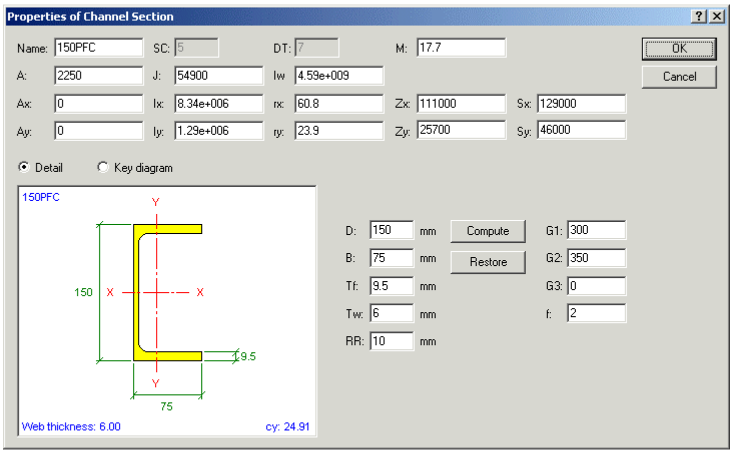


SECTION LIBRARY MANAGER

You may select any library, category, or section on the left and click the arrow button to send it to the destination library on the right. Double-clicking a section on the right will display a dialog box in which you may alter any value.

Section Properties Dialog Box

The properties of any section in the destination library may be displayed by right-clicking the section and choosing Section Properties on the pop-up menu. Double-clicking the section will also display the section properties dialog box. The dialog box shows all the values stored in the library for the section. Any values that are not disabled in the dialog box may be changed. Click the ? button at the top and then click on any item for help. Clicking the Compute button computes all derived values from the current dimensions. The Restore button sets all edit boxes back to their original values.



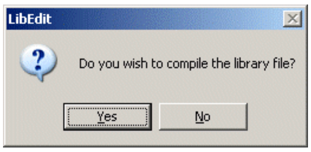
SECTION PROPERTIES DIALOG BOX FOR CHANNEL

Section property dialog boxes for some sections have an Ax, Ay button, which computes shear areas. For an I section Ax is computed as the nett web area and Ay is computed as 5/6 of the flange area. For SHS, RHS, and box sections, Ax is the nett "web" area where the web is considered to include both sides. Similarly, Ay is the nett area of the top and bottom "flanges" – this does not include overhang in the case of the box section.

Note: Shear areas are usually set to zero, causing Microstran to ignore shear distortion.

Compiling a Library

When you click the Save button you can initiate the compilation of the library source file into a Microstran library. Click Yes in the dialog box below to do this.



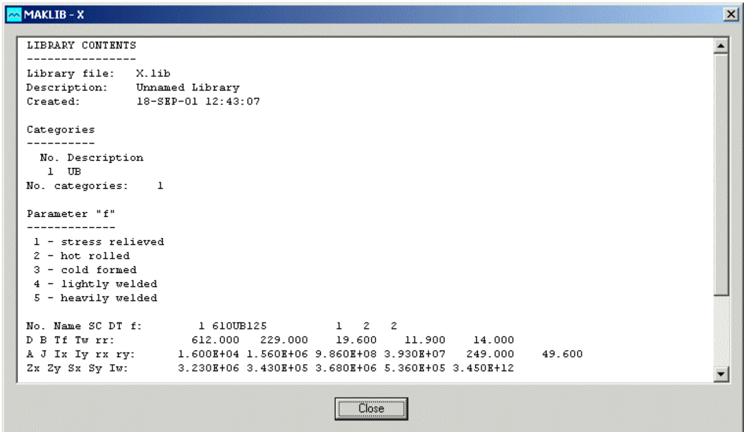
COMPILING THE LIBRARY

The library compiler reads and interprets the library source file and writes a Microstran library file. The value of any section property value input as zero is computed automatically provided sufficient dimensions for the calculation have been input. When compilation has finished successfully a report is displayed, as shown below. This report lists any errors or inconsistencies detected in the input data. The library report for

the current library may be printed or previewed from the Microstran File menu.

All section property values are computed from the section dimensions and where the corresponding input value differs by more than 1%, a warning message results. Section properties are computed only approximately for taper flange sections so warnings for these sections may not be valid.

Note: Microstran and Limcon libraries are compatible.



COMPILING THE LIBRARY – REPORT

The Material Library

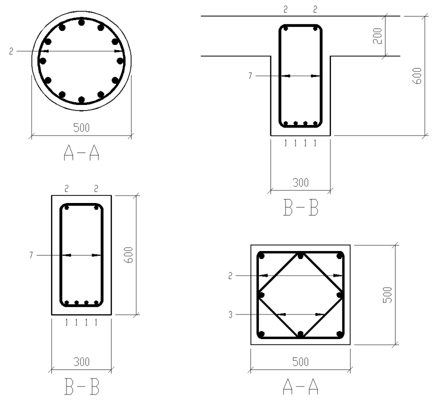
The material library may be edited by selecting the **File > Configure > Edit Library** command. This initiates editing of a library source file in Microstran's text editor. On completion of editing you may elect to compile the source file.

The material property library text file from which the library file Matl.lib is compiled is shown below:

```
2   1
Materials Library  N/mm2, , kg/m3, degC
   4  1.0  1.0  1.0  1.0
$
$ Steel props:    Ref AISC DCT, 1st Edition
$ Concrete props: Ref AS3600
$ Timber props:   Ref AS1720.1
$ File is free format
$
$ <matl name> <Young's mod> <Poisson's ratio> <mass density>
<coeff expansion>
$
STEEL      200E3  0.25 7850. 1.17E-5
ALUMINIUM  65E3   0.33 2700. 2.10E-5
CONC20     25500  0.20 2450. 1.17E-5
CONC25     28600  0.20 2450. 1.17E-5
CONC32     32300  0.20 2450. 1.17E-5
CONC40     36100  0.20 2450. 1.17E-5
CONC50     40400  0.20 2450. 1.17E-5
F5          6900  0.    850. 0.
F11        10500  0.   1050. 0.
F17        14000  0.    650. 0.
F27        18500  0.    650. 0.
```

19:RC Design & Detailing

General



RC design modules require the use of kN and meter units in the structure model. For convenience, section dimensions are in mm..

Reinforced concrete design options for Microstran allow you to design reinforced concrete beams and columns. Design options are available to AS 3600-1994 “Concrete Structures” and BS 8110:Part 1:1985 “Structural Use of Concrete”. The RC design options operate on standard Microstran models so that once you have analysed a structure you may proceed to determine the reinforcement for each member in accordance with the requirements of the design code. Section and reinforcement details may be changed as required, based on the results of design computations.

You must initialize all members to be designed and also select the design load cases before any design operations may be performed. Initializing members assigns default design data that may be displayed or edited with the **Design > Design Data** command. Once entered, this data is saved automatically when you save the Microstran job (in the Job.msw file, where “Job” is the job name), and when you export to an archive file. The design procedure automatically accesses the design data and analysis results for the selected load cases.

If the design indicates different sections to the ones originally specified the model should be updated and the analysis repeated to take account of any resulting redistribution of member forces.

Detailing Option

The reinforced concrete detailing option uses the results generated by the design module to generate a CAD drawing (in standard DXF format) with reinforcement details and bar lists for all beams and columns designed.

Limitations

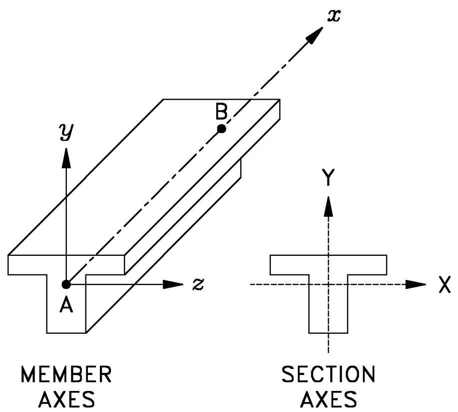
The following limitations apply to the reinforced concrete design module:

- The model must use units of kN and meters.
- Columns are symmetrically reinforced with the same cover on each face.
- Reinforcing bars are not bundled.
- Beams are loaded so as to bend about the member *z* axis (see "Section Axes", below).

For meaningful deflection computations, members should extend from intersection point to intersection point without additional intermediate nodes.

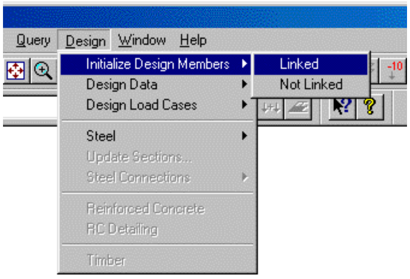
Section Axes

The reinforced concrete design module uses "section" axes. The section X and Y axes are parallel to the member *z* and *y* axes respectively, as illustrated in the accompanying diagram (see also "Coordinate Systems" on page 63). The section axes may also be referred to as XX and YY.



SECTION AXES

Initializing Design Members



DESIGN > INITIALIZE DESIGN MEMBERS

Initializing design members performs three functions:

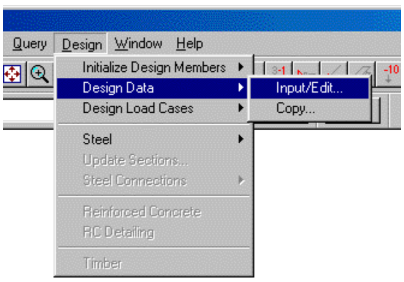
- Associates each initialized member with a specific design code.
- Assigns default design data to initialized members.
- Links members together into design members, where appropriate.

Linking of RC members controls detailing operations. Linked members. will be detailed as a single "run" of members instead of individual beams or columns. Linked members must be collinear, have the same section properties and orientation, and have no member releases or rigid offsets.

Select the members you wish to initialize (see "Selecting Nodes and Members" on page 26).

Default member design data, including the reinforced concrete design code, is specified with the **File > Configure > Reinforced Concrete Design** command. By default, the section type is set to that used in analysis.

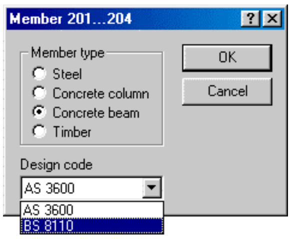
Design Data



DESIGN > DESIGN DATA

On selecting the **Design > Design Data > Input/Edit** command select an initialized member so you can enter design data. If you find the items on the sub-menu are disabled after selecting **Design Data**, select the **Design > Initialize Design Members** command.

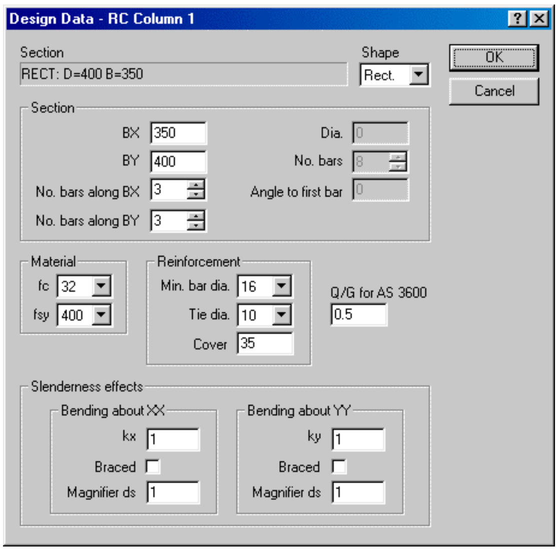
In the first dialog box, you choose the design code from those available in the list box.



CHOOSING THE DESIGN CODE

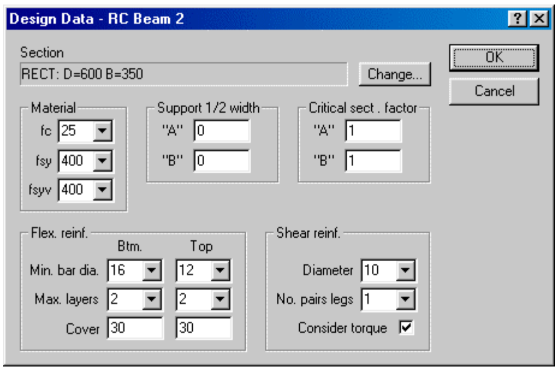
The next step is to enter basic design data for the selected member in one of the dialog boxes shown below. For members initialized as reinforced concrete columns the first dialog box is displayed and for beams the second is displayed. Default data may be modified via options on the **File > Configure** menu.

Tool tips are provided for all items in the design dialog boxes. These are small pop-up windows that appear as the cursor passes over an item. Additional pop-up help may be obtained for any item by clicking the  button at the top of the dialog box and then clicking on the item.



DESIGN DATA – RC COLUMN

The data required for column design is the shape (rectangular or circular), section dimensions, desired bar pattern and clear covers, the effective length, and sway condition. For unbraced columns designed to AS 3600, you must also input the value of the storey moment magnifier δ_s.



DESIGN DATA – RC BEAM

The data required for beam design is the beam shape (rectangular, T, or L), the dimensions of the concrete section, reinforcement data, top and bottom clear covers, widths of the supports at the ends of the beam, locations of the critical sections for shear, and whether the design is to consider the torsion provisions of the design code.

Note that "Support 1/2 width" is the distance from the beam end node to the face of the support. If the beam is supported by columns, this value will normally be half the column width.

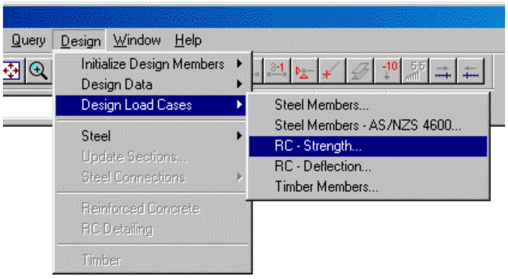
Default Values

The default values for reinforced concrete design and detailing parameters are read from the Msrc.def file in the library folder. Each line in the file contains a keyword commencing in column 1 and one or more items of data. The dollar character "$" is used to indicate that the remainder of the line is a comment, which is ignored by the program. There are different sections in the file for different design codes. The section of the defaults file that refers to AS 3600 is listed below.

```
$
$ Concrete Defaults File MSRC.DEF

$
$ Default data for AS3600
$
RC_DATA AS3600

$
$ Default data for columns:
$
 FCC    32.0       $ f'c
 FSYC  400.0       $ fsy
 DIABC  16.0       $ minimum dia. for longitudinal bars
 DIATC   10.       $ diameter of ties
 CVCOL   35.       $ clear cover to ties
 CSBLK  RECT       $ stress block - CEB/RECT
 KX      1.0       $ effective length factors
 KY      1.0
 XSWAY     Y       $ sway about section X (member z) axis - Y=Yes N=No
 YSWAY     N       $ sway about section Y (member y) axis - Y=Yes N=No

$ Default column shape - comment out whichever is not required:
 CSHAPE RECT 350. 350. 3 3   $ rect. column - bx, by, nx, ny
$ CSHAPE CIRC 400. 8 0.0     $ circ. column - dia., no. bars, start angle

$
$ Default data for beams:
$
 FCB      25.     $ f'c
 FSYB    400.     $ fsy for longitudinal reinforcement
 DIATB    12.     $ minimum dia. top bars
 DIABB    16.     $ minimum dia. bottom bars
 FSYV    400.     $ fsy.f for transverse reinforcement
 CVBEM    30.     $ clear cover to shear reinforcement
 DSHEAR   10.     $ dia. shear reinforcement
 NPLEG     1      $ no. pairs of legs in shear reinf.
 RSHEAR    N      $ shear flag - Y if reinf. carries total shear
                  $              N otherwise
 TORQUE    Y      $ design for torsion
 MLTOP     2      $ max. no. layers, top bars
 MLBTM     2      $ max. no. layers, bottom bars
 CRITVL  1.0      $ dist. section for shear=CRITVL*d from LH support
 CRITVR  1.0      $ dist. section for shear=CRITVR*d from RH support

$ Default beam shape - comment out whichever is not required:
 BSHAPE RECT   450. 350.           $ rect. beam - depth, width
$ BSHAPE TEE   500. 350. 1200. 120. $ tee beam - depth, web width,
                                    $     flange width, flange thickness

 CGRAD   20.  25.  32.  40.  50.    $ concrete grades (up to 10)
 SGRAD   250.  400. 450.            $ reinf. grades (up to 5)
 SDIA   6.  8. 10.  12.  16.  20.  24.  28.  32.   36.  $ reinf. diameters
 SAREA 28. 50. 78. 110. 200. 310. 450. 620. 800. 1020.  $ reinf. areas
                                                        $ up to 15 sizes

 TSPAC   37.5 50. 75. 100. 150. 200. 300. 450. 600. $ trans. reinf.
                                                    $ standard spaces
                                                    $ up to 15 spaces
END
```

Alternative CSHAPE and BSHAPE statements are preceded by the “$” character. Moving the “$” sign changes the default column or beam cross-section.

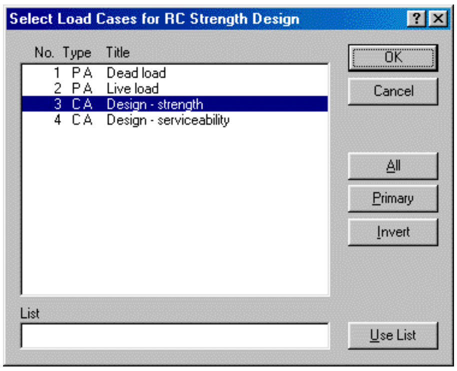
Selecting Design Load Cases



DESIGN > DESIGN LOAD CASES

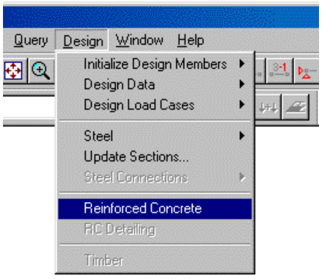
You must select the load cases to be considered in the reinforced concrete design. One or more load cases must be specified for strength design. If load cases are not specified for deflection design, deflection checks are not performed.

All load case titles are shown in a list box and you may select any of these by clicking on them. Sometimes, it might be more convenient to enter a list of load case numbers rather than selecting graphically. To do this, enter the list in the **List** edit box and click the **Use List** button. The rules for lists are set out in Chapter 15 – "Reports".



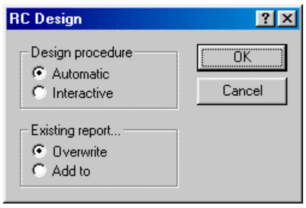
SELECTING LOAD CASES

The Design Process



DESIGN > REINFORCED CONCRETE

The **Design > Reinforced Concrete** command initiates the member design process. Select the members to be designed (see "Selecting Nodes and Members" on page 26). The dialog box below lets you choose whether the design procedure is to be automatic or interactive. You also specify whether an existing design report is to be overwritten or added to (design reports are automatically deleted when you close a job so, initially, there is no existing design report).



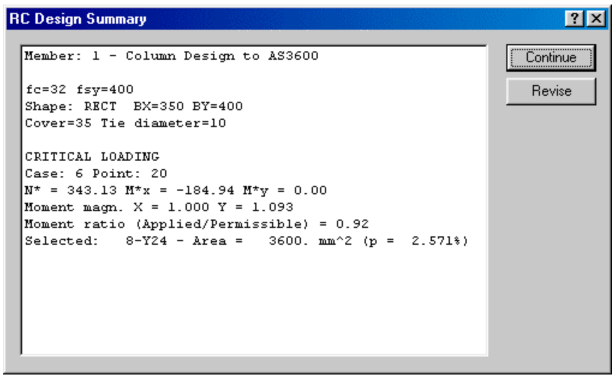
CHOOSING DESIGN PROCEDURE

The reinforcement requirements are determined for each of the selected members. If for any member excessive reinforcement would be required, or if there are no bars large enough, the design data dialog box will be displayed for the current member so you can change the section dimensions as required.

If the interactive design mode is chosen you have an opportunity to review the design. The interactive design procedure is discussed in detail below.

Designing Columns

When you select the interactive design mode the results of the current design are displayed.

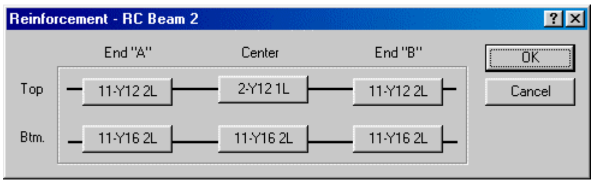


INTERACTIVE DISPLAY – COLUMN DESIGN

If the design details are satisfactory click the **Continue** button to proceed to the next member selected for design. Clicking the **Revise** button allows you to change the column design data and repeat the design.

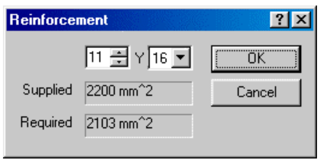
Designing Beams

When you select the interactive design mode the reinforcement layout selected for the beam is displayed.



REINFORCEMENT LAYOUT

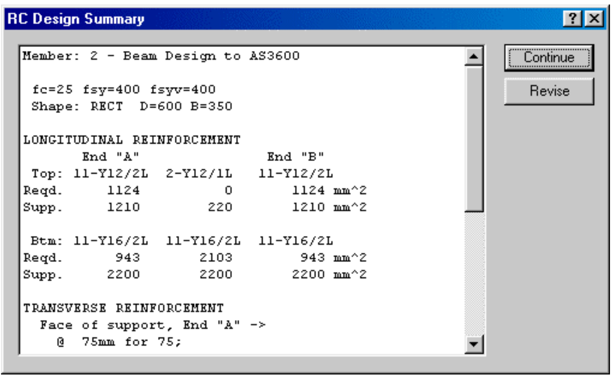
The bar arrangement is shown on a button at each of 6 locations in the beam. The notation “11-Y16 2L”, for example, denotes 11 × Y16 bars in 2 layers. You may click any of these buttons to change the bar arrangement at the corresponding location in the beam. If you do so, the cross-sectional area of the selected arrangement is displayed, together with the area of steel required by the design.



CHANGING BAR ARRANGEMENT

If you are going to use the detailing option you should ensure that a single bar size is used at all bottom locations in any design member.

The results of the current design are then displayed.



INTERACTIVE DISPLAY – BEAM DESIGN

If the design details are satisfactory click the **Continue** button to proceed to the next member selected for design. Clicking the **Revise** button allows you to change the column design data and repeat the design.

Obtaining Design Results

After designing members the design results may be displayed or reported in a number of ways:

Use the **Query > Design Member** command to show the design status in the output window for any selected member.

Design results are automatically written to a design report file during reinforced concrete design. The design report may be previewed with the **File > Print Preview** command and it may be printed with the **File > Print File** command. Note that there are extensive facilities for formatting the design report using the **File > Page Setup** command.

RC design results are not saved when you save a job and the RC design report file is automatically deleted (if it exists) when the job is closed. The RC design report may be recreated by repeating the design procedure.

The RC design report file is created in the data folder and it is named Job.p5, where “Job” is the job name. You may save an RC design report file by dragging it to another folder using Windows Explorer.

Column Design Computations

Microstran checks the column section with the minimum reinforcement size in the specified pattern along the full length of member. If the column strength is inadequate at any point larger bars will be selected and bar positions adjusted to maintain cover before checking proceeds.

The following general assumptions are made in computing the failure load:

- The strain distribution in both concrete and reinforcement is linear across the section, i.e. plane sections remain plane.
- The tensile strength of the concrete is neglected.
- The reinforcement is perfectly elastic-plastic with equal tensile and compressive yield stresses.

For AS 3600:

Bending moments are magnified if the column is slender. If the column is unbraced, the moment magnifier is taken as the maximum of δ_s (entered by the designer) and the computed value, δ_b. The minimum moment of 0.05D × N* is applied.

The stresses in the concrete are described by either the CEB curve or a rectangular stress block, depending on the setting of the stress block flag in the default data file. The ultimate strain in the concrete is 0.003 when the neutral axis is within the section and 0.002 when the neutral axis is at infinity. Intermediate values are obtained by linear interpolation.

For BS 8110:

Additional moments induced by lateral deflection of the column at the ultimate limit state are computed in accordance with Clause 3.8.3. A minimum design moment due to the axial force acting at an eccentricity of 0.05 times the dimension of the column in the plane of bending, up to a maximum of 20 mm, is also applied.

The stresses in the concrete are described by the simplified stress block of BS 8110 Fig. 3.3. The ultimate strain in the concrete is 0.0035.

General

A rigorous method is used to calculate biaxial bending effects. It does not use empirical interaction formulae in conjunction with uniaxial capacities. Capacities are computed using the area and coordinates of each bar and not by using an “equivalent” steel area. As the computations proceed, the positions of the reinforcing bars are adjusted to reflect the largest size so far determined.

On completion of the bar area calculations a bar size that satisfies the most critical load condition is selected.

The position of the neutral axis (as an offset and angle from the plastic centroid) and the maximum eccentricity are then determined for each load condition. The maximum eccentricity is that at which the axial force causes the section to reach its effective strength when applied at an angle

consistent with the ratio of the moments (adjusted for slenderness) about the section X and Y axes. The difference between the load and maximum eccentricities gives a measure of the excess capacity of the section. The excess is due to the difference between the computed and selected bar areas.

In interactive mode, you may accept the design or revise the section dimensions and reinforcement patterns. On acceptance, a design report for the member is added to the design report file.

Beam Design Computations

Longitudinal reinforcement is determined from bending moments about the section X axis at the face of the supports. Bar patterns are selected at the ends and centre of the beam and checked for clearance. If clearance is inadequate, effective depths are adjusted and steel areas are recomputed. Compression reinforcement is added as needed. Torsional steel requirements (longitudinal reinforcement) are computed assuming a “rational” layout of the flexural reinforcement. Bending moments are not redistributed.

Member forces at the face of the support are approximated by interpolation. This procedure is not accurate if the support width is large – in this case, the structural model should include rigid member offsets to obtain accurate member forces.

In interactive mode, you may modify the longitudinal reinforcement selected by the program to match that in adjacent beams or to control deflections.

Axial forces, and moments about the section Y axis, are not considered in the computation of longitudinal reinforcement for beams. If these actions are significant you will need to carry out additional manual calculations.

Transverse reinforcement is computed for each load case in turn and the minimum bar spacing at each point is used to select an arrangement of links based on a set of preferred spacings. Anchorage of shear reinforcement is not checked.

Deflections are computed by scaling the analysis deflections by the ratio of the stiffness (EI) used in the analysis to that computed for the selected section. Effective inertias are computed for all cases designated for deflection from Branson’s formula:

$$I_{ef} = I_{cr} + \left(I - I_{cr}\right)\left(\frac{M_{cr}}{M_s}\right)^3$$

The lowest value of I_{ef} is then used in all deflection calculations.

The deflections reported are the maximum displacements in the member *y* direction measured from the displaced position of the chord joining the end nodes of the beam. For cantilever ends – assumed for nodes with no support in the direction of the member *y* axis and no connected members

– the tip displacement is reported relative to the displacement of the fixed end.

Immediate deflection and additional long-term deflection (due to creep and shrinkage) are reported for each load case selected for deflection computations. The additional long term deflection is computed by multiplying the short term deflection by the multiplier k_{cs}, defined in AS 3600 Clause 8.5.3.3 as a function of A_{sc}/A_{st}.

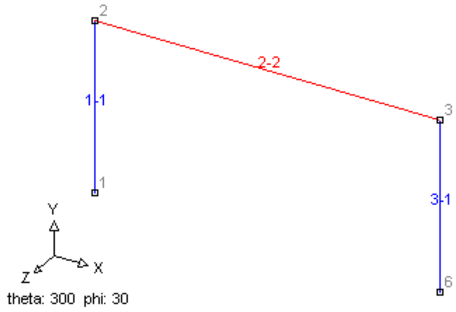
The computed values of gross, cracked and effective inertias and the ratios of compression reinforcement to tensile reinforcement are written to the report.

In interactive mode you have the option of revising the member design. When the design is accepted a report for the member is added to the design report file.

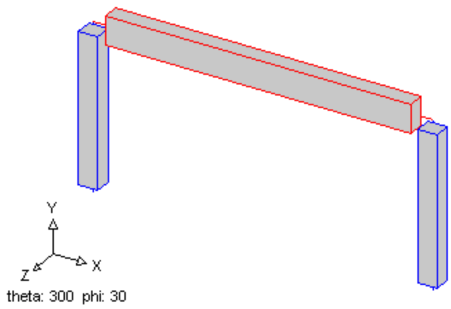
RC Design Example

A small rectangular portal frame has been used as the design example. The archive file defining the job is shown below together with plots of the structure geometry, applied loads and bending moment for the design load cases.

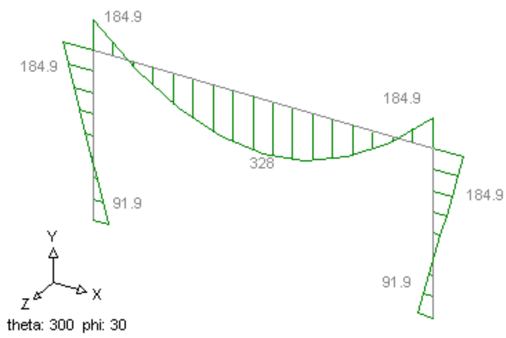
```
* TEST FRAME FOR RC DESIGN
* SINGLE BAY RECTANGULAR PORTAL FRAME
*
*
VERS     4
TYPE     3
VERT     2
UNIT     1 m   kN  T   C

NODE     1     0.0000     0.0000     0.0000 111111
NODE     2     0.0000     3.5000     0.0000 001110
NODE     3     7.0000     3.5000     0.0000 001110
NODE     6     7.0000     0.0000     0.0000 111111

MEMB     1     1     2     D     1     1    000000 000000
MEMB     2     2     3     D     2     1    000000 000000
MEMB     3     6     3     D     1     1    000000 000000

PROP     1 SHAP RECT COLUMN          400x350
      0.400     0.350      1.00
PROP     2 SHAP RECT RAFTER          600x350
      0.600     0.350      1.00

MATL     1 3.230E+07 2.000E-01 2.450E+00 1.170E-05

CASE     1 DEAD LOAD
NDLD     2      0.000    -40.000      0.000      0.000      0.000      0.000
NDLD     3      0.000    -40.000      0.000      0.000      0.000      0.000
MBLD     2  UNIF  FY  GL          -25.000

CASE     2 LIVE LOAD
MBLD     2  UNIF  FY  GL          -35.000

CASE     3 WIND LOAD
MBLD     1  UNIF  FY  LO           -3.600
MBLD     2  UNIF  FY  LO           -4.900
MBLD     3  UNIF  FY  LO            2.300

CASE     4 G + 0.7Q
COMB     1      1.000
COMB     2      0.700

CASE     5 1.25G + 1.5Q
COMB     1      1.250
COMB     2      1.500

CASE     6 1.25G + 0.4Q + 1.5WU
COMB     1      1.250
COMB     2      0.400
COMB     3      1.500

END
```



DESIGN EXAMPLE



DESIGN EXAMPLE – SECTIONS



DESIGN EXAMPLE – BENDING MOMENT

Design Example – Report File

```
== L O A D   C A S E S   -   R C   S T R E N G T H   D E S I G N ==

  Case Type  Title
     5   C   1.25G + 1.5Q
     6   C   1.25G + 0.4Q + 1.5WU


== L O A D   C A S E S   -   R C   D E F L E C T I O N   C H E C K ==

  Case Type  Title
     4   C   G + 0.7Q


== R C   C O L U M N   D E S I G N   R E P O R T ==

MEMBER: 1     (Design to AS 3600)

Node "A":     1         Node "B":     2         Ref. node/axis:    -X
     f'c:    30 MPa          fsy:   400 MPa                Q/G:  0.00
Tie dia.:    12 mm         Cover:    40 mm
   Shape:  RECT            Width:   350 mm  sect. "x" dim. (|| member "z")
                           Depth:   400 mm  sect. "y" dim. (|| member "y")
                   Reinforcement:     3 bars each face || member "z" axis
                         Pattern:     3 bars each face || member "y" axis

  Length:   3.5 m             kx:   1.00              ky:   1.00
                             Lex:    3.5 m           Ley:    3.5 m
                              rx:    120 mm           ry:    105 mm
                          Braced:    Yes          Braced:    Yes
                      Eff. depth:    282 mm   Eff. depth:    242 mm
                         i.Mubx:  189.8 kNm       i.Muby:  158.2 kNm

 Case     N*   á.dx     Ncx   kmx   ë.bx     á.dy     Ncy   kmy   ë.by
    5    343      -- short column --        1.000    3645 1.000  1.104
    6    234      -- short column --        1.000    3645 1.000  1.068
    á.dx = ád for XX bending (10.4.3)
    á.dy = ád for YY bending
    ë.bx = Braced column moment magnifier for XX bending
    ë.by = Braced column moment magnifier for YY bending

Member Force Ranges:      N:     -343.12      -233.58 kN  (Compression -ve)
(Analysis)               Vy:       37.43        79.09 kN  (Shear)
                         Vx:        0.00         0.00 kN  (Shear)
                          T:        0.00         0.00 kNm (Torque)
                         My:        0.00         0.00 kNm (Moment)
                         Mx:     -184.94        91.87 kNm (Moment)

Maximum Load Conditions:
 Case   Pt.        N*        M*x        M*y    Magn.x    Magn.y
                   kN        kNm        kNm
    5   21     343.12    -184.94       0.00     1.000     1.104

 Magnified moment resultant is 0.970 of ultimate moment resultant for
 an axial load of 343.12 kN.

*** Selected:   8-Y24 - Reinforcement percentage:  2.571%
                        (Rectangular stress block)


== R C   B E A M   D E S I G N   R E P O R T ==

MEMBER: 2     (Design to AS 3600)

Length:  7.0 m
Node "A":     2         Node "B":     3         Ref. node/axis:     Y
                Shape: RECT        Web width:  350 mm        Depth:  600 mm
         Stirrup dia.:   12 mm     Top cover:   40 mm   Btm. cover:   40 mm
               Support half-width -     Left:  225 mm        Right:  225 mm
                  f'c:   32 MPa          fsy:  400 MPa       fsy.f:  400 MPa

Member Force Ranges:      N:      -79.09       -56.33 kN  (Compression -ve)
(Analysis)               Vy:     -293.12       293.12 kN  (Shear)
                         Vx:        0.00         0.00 kN  (Shear)
                          T:        0.00         0.00 kNm (Torque)
                         My:        0.00         0.00 kNm (Moment)
                         Mx:     -184.94       328.02 kNm (Moment)
```

```
-- Longitudinal Design --

                     End "A"                               End "B"
     Mxx (neg.):      141.1               0.0               141.1
  Reinf. (neg.):   3-Y20 1L          2-Y20 1L            3-Y20 1L
    Area (mm^2):        930               620                 930
     Percentage:     0.494%            0.329%              0.494%

     Mxx (pos.):      143.4             328.0               143.4
  Reinf. (pos.):   7-Y20 2L          7-Y20 2L            7-Y20 2L
    Area (mm^2):       2170              2170                2170
     Percentage:     1.168%            1.168%              1.168%

Code minimum strength (8.1.4.1):
     M*min.neg:      68.4 kNm    As:      404. mm^2
     M*min.pos:      68.4 kNm    As:      404. mm^2

         Neg.         Pos.          Flex.
     /-Bending-/  /-Bending-/  /--Required-/  /----Torsion----/  /--Supplied-/
Pt.    ku    M*     ku    M*   Astop  Asbtm    T*  Astop Asbtm   Astop  Asbtm
 1   0.08  141.  0.00     0.    848      0     0.     0     0     930   2170
 2   0.05   87.  0.00     0.    519      0     0.     0     0     930   2170
 3   0.04    1.  0.00     0.    404      0     0.     0     0     930   2170
 4   0.00    0.  0.04    77.      0    460     0.     0     0     930   2170
 5   0.00    0.  0.08   143.      0    875     0.     0     0     930   2170
 6   0.00    0.  0.12   200.      0   1238     0.     0     0     620   2170
 7   0.00    0.  0.15   246.      0   1544     0.     0     0     620   2170
 8   0.00    0.  0.17   282.      0   1788     0.     0     0     620   2170
 9   0.00    0.  0.19   308.      0   1966     0.     0     0     620   2170
10   0.00    0.  0.20   323.      0   2075     0.     0     0     620   2170
11   0.00    0.  0.20   328.      0   2111     0.     0     0     620   2170
12   0.00    0.  0.20   323.      0   2075     0.     0     0     620   2170
13   0.00    0.  0.19   308.      0   1966     0.     0     0     620   2170
14   0.00    0.  0.17   282.      0   1788     0.     0     0     620   2170
15   0.00    0.  0.15   246.      0   1544     0.     0     0     620   2170
16   0.00    0.  0.12   200.      0   1238     0.     0     0     620   2170
17   0.00    0.  0.08   143.      0    875     0.     0     0     930   2170
18   0.00    0.  0.04    77.      0    460     0.     0     0     930   2170
19   0.04    4.  0.00     0.    404      0     0.     0     0     930   2170
20   0.05   87.  0.00     0.    519      0     0.     0     0     930   2170
21   0.08  141.  0.00     0.    848      0     0.     0     0     930   2170

-- Transverse Design --

 2 Legged stirrups - Dia: 12 mm   Grade: 400 MPa
Concrete carries part of shear.
Critical sections at:  1.00*d from face of left support.
                       1.00*d from face of right support.
Shear at critical sections - Left:    -229.2 kN   Right:     229.2 kN

    Tuc:     49.9 kNm
 Tu.max:    188.2 kNm

Pt.  Case     V*       T*       N*    Ast      Vuc       s
              kN      kNm       kN   mm^2       kN      mm
 1     5    229.       0.      79.    930      117.    300
 2     5    229.       0.      79.    930      117.    300
 3     5    229.       0.      79.    930      117.    300
 4     5    205.       0.      79.   2170      156.    300
 5     5    176.       0.      79.   2170      156.    450
 6     5    147.       0.      79.   2170      156.    450
 7     5    117.       0.      79.   2170      156.    450
 8     5     88.       0.      79.   2170      156.    450
 9     5     59.       0.      79.   2170      156.    450
10     5     29.       0.      79.   2170      156.  99999
11     6      1.       0.      56.   2170      156.  99999
12     5     29.       0.      79.   2170      156.  99999
13     5     59.       0.      79.   2170      156.    450
14     5     88.       0.      79.   2170      156.    450
15     5    117.       0.      79.   2170      156.    450
16     5    147.       0.      79.   2170      156.    450
17     5    176.       0.      79.   2170      156.    450
18     5    205.       0.      79.   2170      156.    300
19     5    229.       0.      79.    930      117.    300
20     5    229.       0.      79.    930      117.    300
21     5    229.       0.      79.    930      117.    300
```

```
Selected Stirrup Spacings:
  Face of support, End "A" ->
                      @ 75 mm for   75 mm;
                      @300 mm for 1200 mm;
                      @450 mm for 1800 mm;
                      @250 mm for  250 mm;
                      @450 mm for 1350 mm;
                      @300 mm for 1800 mm;
                      @ 75 mm for   75 mm;
  Face of support, End "B" ->

-- Deflections --

Deflections are measured in local "y" direction.
  Cantilevers - relative tip deflection.
  Beams - relative to displaced position of end nodes.

Load case giving minimum Ieff:
Case     4                       Left         Centre         Right
                        M:      -109.3         193.9        -109.3 kNm
                      Mcr:        71.3          71.3          71.3 kNm
                      Igr:   6.300E+09     6.300E+09     6.300E+09 mm^4
                      Icr:   1.336E+09     2.570E+09     1.336E+09 mm^4
                     Ieff:   2.712E+09     2.755E+09     2.712E+09 mm^4
                 NA depth:       110.4         168.9         110.4 mm
                  Asc/Ast:       2.333         0.286         2.333

    Effective Inertia Ieff:   2.734E+09 mm^4

           /---- Immediate ---/     Addn. Long Term
 Case      Defl. mm     L/defl.     Defl. mm
    4        -10.9         643        -18.0
```

Reinforced Concrete Detailing Option

The integrated reinforced concrete detailing option may be used with any of the Microstran reinforced concrete design modules. After a reinforced concrete design session you may select the **Design > RC Detailing** command and the detailing module will automatically produce a CAD transfer file containing the reinforcement layout for all beams and columns just designed. Detailing generally follows standard industry practice as described in *Standard Method of Detailing Structural Concrete* (Institution of Structural Engineers, UK), and the *Reinforcement Detailing Handbook* (Concrete Institute of Australia).

Limitations

The following limitations apply to the reinforced concrete detailing module:

- The Microstran model has either the Y or the Z global axis vertical.
- Beams are parallel to a global axis and columns are vertical.
- Members to be detailed in a single run must have been initialized as linked.
- Columns must have their local x axis up; i.e. the “A” node must be at the bottom.

Operation

Reinforced concrete details are generated by selecting the **Design > RC Detailing** command after reinforced concrete design has been

successfully completed for one or more members. Reinforcement details are generated in a standard format DXF file (AutoCAD R12 format) named Job.dxf, where “Job” is the current job name. This file may be read into most CAD programs (e.g. using the DXFIN command in AutoCAD).

The resulting CAD drawing contains all beam and column details for the last design session. These details may then be transferred as required to reinforcement detail sheets. *It is intended that the details produced will be reviewed by the designer* with any inconsistencies being remedied as required. Warning messages (on a separate layer) will appear on the drawing when known unresolved situations occur.

Drawing Layout

The CAD drawing is full-size in millimeters with cross-sections drawn at double scale. Beams are drawn on the left, starting at the bottom of the drawing. Cross-sections are shown below the beam for each mid-span and support location. Where possible, beams are drawn so that grid lines are aligned vertically. A reinforcement schedule is generated for each run of beams with bar data, including bar lengths, summarized in tabular form. Columns are drawn starting to the right of the beams. An elevation is shown for each column with a mid-height cross-section shown on the right for each storey. Only one main bar is drawn in the elevation but all are shown in the cross-section. All column elevations are shown from the same direction; either looking in the -Z or -X directions for vertical Y axis, or the -Y or -X directions for vertical Z axis. Beams framing into the column are shown to scale in elevation and section. A reinforcement schedule is also shown for each run of columns.

Input

The reinforced concrete detailing module obtains both geometric and design data directly from Microstran. CAD drawing defaults are defined in the Msrc.def file and the styles to be used for text and dimensions are specified in a header DXF file.

CAD Drawing Defaults

The default values used by the program may be changed by selecting the **File > Configure > RC Design > Default RC Data** command. This command initiates Microstran's text editor, displaying the Msrc.def file. The part of the file that affects detailing is shown below. It defines drawing parameters such as layer names and configurable detailing parameters.

Each line in the file contains a keyword followed by a parameter value. Anything following the dollar symbol is a comment.

```
$
$ Default data for detailing (AS)
$
DETAIL_DATA AS3600
 DXF_HEAD        HEADER.DXF  $ Name of header DXF - don't change
 LAYER_CL        CENTRE      $ Name of layer for centre-lines
 LAYER_CONC      OUTLINE     $ Name of layer for concrete outline
 LAYER_REINF     REINF       $ Name of layer for reinforcement
 LAYER_TEXT      TEXT        $ Name of layer for text
 LAYER_DIMS      DIMENS      $ Name of layer for dimensions
 LAYER_LINKDIMS  LINKDIMENS  $ Name of layer for link dimension lines
 LAYER_SUPPS     SUPPORTS    $ Name of layer for supports
 LAYER_MSG       MESSAGES    $ Name of layer for messages
 TEXT_STYLE      STANDARD    $ Name of text style
 BEAM_HCLEAR     3500        $ Horizontal clearance - beams
 BEAM_VCLEAR     10000       $ Vertical clearance - beams
 COL_CLEXTABOVE  920         $ Extension of column centre-line above beam
 COL_CLEXTBELOW  780         $ Extension of column centre-line below beam
 BEAM_DIMABOVE   1600        $ Start distance for dimensioning above beam
 BEAM_DIMBELOW   800         $ Start distance for dimensioning below beam
 BEND_LENGTH     100         $ Length of bar end bend
 BEND_OFFSET     25          $ Lateral offset of bar end bend
 COL_HCLEAR      6000        $ Horizontal clearance - columns
 COL_VCLEAR      1000        $ Vertical clearance - columns
 XSECT_FACT      2.00        $ Scale factor for plotting cross-sections
 GRADE_SEP       300         $ Transition steel grade
 GRADE_BELOW     R           $ Bar type for steel grade below transition grade
 GRADE_ABOVE     Y           $ Bar type for steel grade above transition grade
 COL_ELEV        1           $ Column elevation: 1=Normal 2=Rotated
 LINKS_DIAMOND   1           $ Use diamond-shaped links: 1=Yes 0=No
END
```

Note that:

- All layers specified in the parameter file must be defined in the header DXF.
- Concrete outlines are drawn as polylines of 5 units width. Supports will be shown if they exist where columns do not.
- Each reinforcing bar is drawn to scale as a polyline with a width equal to the bar size. Reinforcement schedules are drawn in the text layer.
- Where a problem has been encountered in detailing, a warning message is plotted in the messages layer.
- The dimensioning style is determined by the variables set in the header DXF. Any line type, for example, may be changed by changing the line type used for the corresponding layer.
- The specified text style is used for all the drawing except the reinforcement schedules, which are drawn in style MONO. Both text styles must exist in the header DXF and the specified style must

not be MONO. MONO should be defined as a monospaced font so that the columns of text in the schedule are properly aligned.

- The horizontal clearance is the distance in drawing units between the end of a run of beams and the corresponding reinforcement schedule. The same clearance is used between the rightmost beam schedule and the first column detail. The vertical clearance is the distance in drawing units between each run of beams. Horizontal and vertical clearances are also specified for columns.
- The column centre-line extensions above and below the beam are the distances in drawing units that the centre-line extends beyond the beam profile.
- The start distances for dimensioning are the distances in drawing units of the topmost dimension away from the beam profile.
- The length and lateral offset of the bar end bend define the size in drawing units of the artificial bend shown at the ends of horizontal bars to distinguish them from other bars in the same layer.
- The transition steel grade is the grade above which any reinforcement will be assumed to be high-yield deformed bar. At and below this grade, reinforcement will be assumed to be mild steel plain bar. The bar types for steel grade are alphabetical codes prefixed to bar sizes to indicate bar grade, usually “T” or “Y” for high-yield bars and “R” for plain bars.

The Header DXF

Drafting defaults such as line types, dimension style, and text size are specified in the header DXF. This is a file called Header.dxf that is located in the library folder. It may be altered as required by editing the file directly or by reading it into a CAD system then changing the settings and saving the file in the correct format.

The header DXF is, in effect, a prototype drawing to which the reinforced concrete details are added. It is read by the detailing program, interpreted as required, and written to the output DXF. Several style features of the output drawing may be changed automatically by editing this file. For example, the colours or line types of the various layers may be changed in this way. *To avoid the possibility of conflicts between the prototype drawing and the header DXF, it is recommended that your CAD program is configured for no prototype drawing.*

The Header.dxf file distributed with the detailing module is compatible with AutoCAD Release 10, 11, or 12, and also AutoCAD LT. This file may be changed as required, either by editing it directly or by bringing it into the CAD program (DXFIN), making changes, and then outputting it (DXFOUT). Items that may be changed include layer names, colours, and line types, the dimensioning variables, text style and height, and so on. All layers referenced in the Msrc.def file should exist in the header DXF. If arrowheads are required on dimension lines instead of 45° tick marks, the variable DIMTSZ must be set to zero. The header file must

contain the text style referred to in the parameter file and MONO, in that order. The specified style must not be MONO.

The following dimension variables defined in the header DXF are interpreted by the detailing program. If these are altered, the detail drawing will be affected correspondingly.

DIMASZ	Arrow size
DIMDLE	Dimension line extension
DIMDLI	Increment between continuing dimension lines
DIMEXE	Extension distance for extension lines
DIMEXO	Offset distance for extension lines
DIMTSZ	Tick size (arrows if zero)
DIMTVP	Text vertical position
DIMTXT	Text size

Detailing Features

Automatic design and detailing of beams and columns is not a complete substitute for the normal procedures. It can, however, relieve the designer and detailer of much of the routine work, allowing them to concentrate on the aspects that are impossible or difficult for a program such as this to handle. It is emphasized that the designer and detailer are responsible for refining or correcting the automatically produced details. Some areas where attention may be required are:

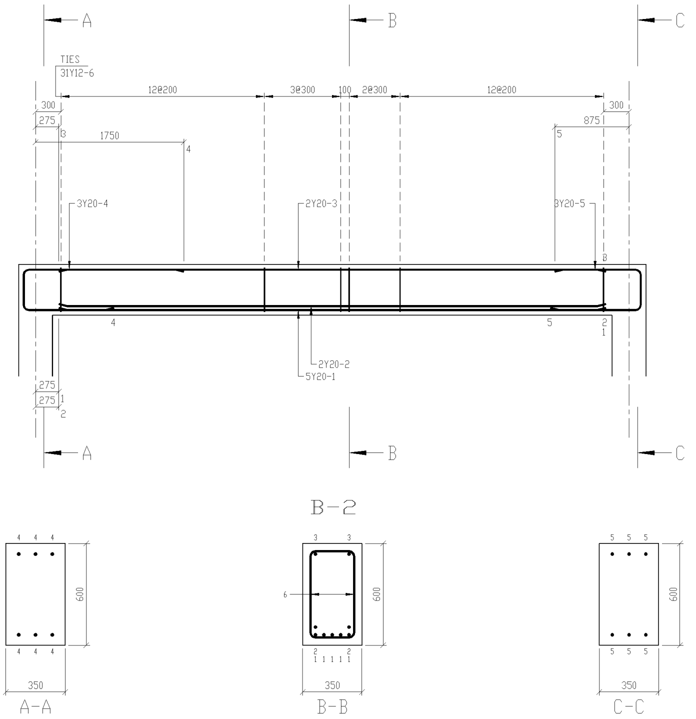
- Link sets in beams are identical throughout the length of the beam (but the spacing may vary). No more than three pairs of legs are permitted in the current version of the software.
- In beams, identical bars in different layers are allocated different numbers.
- At column/beam intersections, no consideration is given to the possibility of interference between beam and column reinforcing bars (or the bars of intersecting beams).
- Beam cross-sections do not show the lapping of the cage steel with bars that are continuous through supports.
- Splice bars in the bottom of beams at supports are terminated 35 bar diameters from the face of the column or support.
- The bottom steel in each beam is constant throughout. Some steel may be saved by specifying cut-off points towards the ends of the span. If you do this, you should be aware that the deflection calculations in the design report may no longer be valid.
- At the top of a run of columns, the main bars are arbitrarily bent inwards, below the structural floor level, in a 90° bend of 8 bar diameters. In some cases, these bars should be bent into adjacent

beams. The designer and detailer must consider what is to be done here.

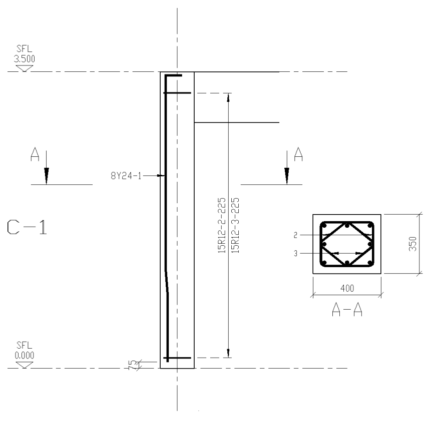
- Starter bars projecting into columns above are terminated 30 bar diameters above the kicker (assumed to be 75 mm high).
- Minimum joggle length in column bars is 300 mm or 10 bar diameters, whichever is the greater.
- There is a 75 mm clearance above starter bars to the start of a joggle.
- Design codes may permit alternate outer vertical bars in columns to be unsupported by links as long as the spacing does not exceed 150 mm. In the current version of the software all vertical bars are restrained by links.
- Diamond-shaped ties may be used instead of single links where there is a central bar in each face of a rectangular column and the maximum included angle does not exceed 135°. A parameter may be set in the Msrc.def file if you do not want diamond-shaped links.

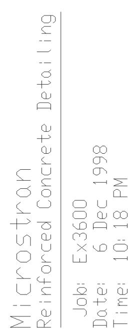
Detailing Example

A small rectangular portal frame has been used as the detailing example. The archive file defining the job is shown in this chapter (see “RC Design Example” on page 330).



BEAM MARK	SPAN LENGTH	BAR NO.	BAR SIZE	BAR LOCN	LAYER NO.	NO. BARS	BAR SHAPE	DIM "A"	DIM "B"	DIM "C"
B-2	7000	1	Y20	BTM	1	5	S	6450		
		2	Y20	BTM	2	2	S	6450		
		3	Y20	TOP	1	2	S	6450		
		4	Y20	B/T	1	3	LL	1898	1073	496
		5	Y20	B/T	1	3	LL	1023	1073	496
		6	Y12	LINK	-	31	T	520	270	





COLUMN MARK	HEIGHT	BAR NO.	BAR SIZE	NO. BARS	BAR SHAPE	DIM "A"	DIM "B"	DIM "C"	DIM "D"	DIM "E"	DIM "F"
C-1	3500	1	Y24	8	CC	795	300	2290	48	3385	192
		2	R12	15	T	320	270				
		3	R12	15	DT	320	270	209			

DETAILING EXAMPLE

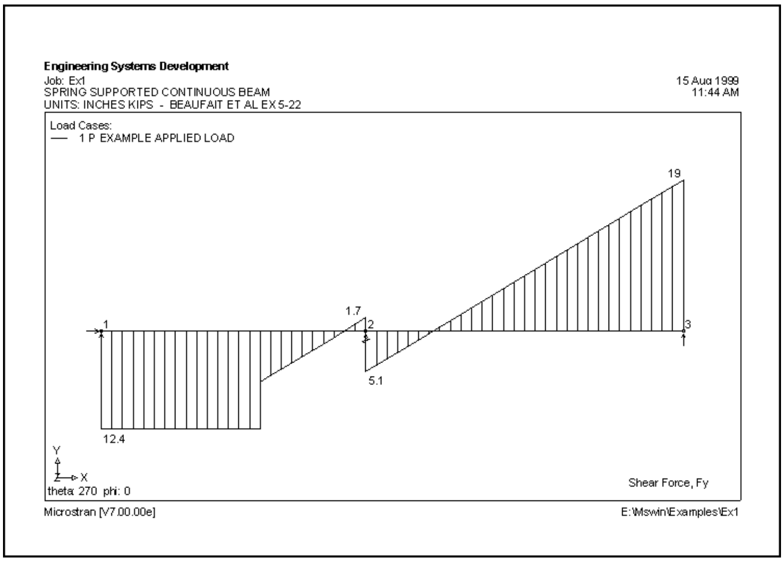
20:Examples

The examples in this chapter are presented to illustrate specific features of Microstran. Examples have been chosen to include several common problem areas in modelling, especially node restraint conditions and member releases.

For all examples the archive file is presented as a complete record of the input data. It is possible to enter data by creating the archive file but it is not usually the most convenient input method. Many common structures can be described easily with Standard Structures Input (see Chapter 8). Report presentations have been varied to demonstrate different features of the report generator. A wide variation of formats is possible by modifying the default values in the Page Setup dialog box.

Example 1 – Continuous Beam

Example 1 is a two-span continuous beam with a central spring support. The structure and shear force diagram are shown in the accompanying output. This is typical of the graphical output that may be produced by Microstran.



EXAMPLE 1 – CONTINUOUS BEAM

A full analysis report has been generated for this structure using the default settings to provide tables of all structure data, loads, member forces, and displacements at the nodes. This is reproduced in the following pages.

Archive File

```
* SPRING SUPPORTED CONTINUOUS BEAM
* UNITS: INCHES KIPS  -  BEAUFAIT ET AL EX 5-22
*
*
VERS    4
TYPE    3
VERT    2
UNIT    5 in  kip k/g F

NODE    1      .0000      .0000      .0000 111110
NODE    2   300.0000      .0000      .0000 001110
NODE    3   660.0000      .0000      .0000 011110

MEMB    1     1     2     D     1     1    000000 000000
MEMB    2     2     3     D     1     1    000000 000000

SPRN    2 0.000E+00 1.000E+01 0.000E+00 0.000E+00 0.000E+00 0.000E+00

PROP    1 PRIS SECT_1
     20.00      .0000      .0000      .0000      .0000      6000.

MATL    1 3.000E+04 3.000E-01 0.000E+00 0.000E+00

CASE    1 EXAMPLE APPLIED LOAD
MBLD    1  CONC  FY  GL  LE       -6.000    180.000
MBLD    1  TRAP  FY  GL  LE        -.067    180.000      -.067    300.000
MBLD    2  UNIF  FY  GL            -.067

END
```

INPUT/ANALYSIS REPORT

```
Job:    Ex1

Title:  SPRING SUPPORTED CONTINUOUS BEAM
        UNITS: INCHES KIPS  -  BEAUFAIT ET AL EX 5-22
Type:   Plane frame
Date:   15 Aug 1999
Time:   11:51 AM

Nodes ............................      3
Members ..........................      2
Spring supports ..................      1
Sections .........................      1
Materials ........................      1
Primary load cases ...............      1
Combination load cases ...........      0

Analysis:  Linear elastic
```

LOAD CASES

```
  Case Type Analysis Title
     1   P      L    EXAMPLE APPLIED LOAD

Analysis Types:
  S - Skipped (not analysed)
  L - Linear
  N - Non-linear
```

NODE COORDINATES

Node	X in	Y in	Z in	Restraint
1	0.000	0.000	0.000	111110
2	300.000	0.000	0.000	001110
3	660.000	0.000	0.000	011110

SPRING SUPPORTS

Node	KX kip/in	KY kip/in	KZ kip/in	KRX in.kip/r	KRY in.kip/r	KRZ in.kip/r
2	0.000E+00	1.000E+01	0.000E+00	0.000E+00	0.000E+00	0.000E+00

MEMBER DEFINITION

Member	A	B	C	Prop	Matl	Rel-A	Rel-B	Length in
1	1	2	Y	1	1	000000	000000	300.000
2	2	3	Y	1	1	000000	000000	360.000

SECTIONS INPUT BY PROPERTY VALUES

Section	Name	Comment
1	SECT_1	

SECTION PROPERTIES

Section	Ax in2	Ay in2	Az in2	J in4	Iy in4	Iz in4	fact
1	2.000E+01	0.000E+00	0.000E+00	0.000E+00	0.000E+00	6.000E+03	

MATERIAL PROPERTIES

Material	E kip/in2	u	Density k/g/in3	Alpha /deg F
1	3.000E+04	0.3000	0.000E+00	0.000E+00

TABLE OF QUANTITIES

MATERIAL 1

Section	Name	Length in	Mass kip/g	Comment
1	SECT_1	660.000	0.000	
		660.000	0.000	

CONDITION NUMBER

Maximum condition number: 2.126E+00 at node: 3 DOFN: 6

APPLIED LOADING

CASE 1: EXAMPLE APPLIED LOAD

Member Loads

Member	Form	T	A	S	F1	X1	F2	X2
1	CONC	FY	GL	LE	-6.000	180.000		
1	TRAP	FY	GL	LE	-0.067	180.000	-0.067	300.000
2	UNIF	FY	GL		-0.067			

Sum of Applied Loads (Global Axes):
FX: 0.000 FY: -38.160 FZ: 0.000

NODE DISPLACEMENTS

CASE 1: EXAMPLE APPLIED LOAD

Node	X-Disp in	Y-Disp in	Z-Disp in	X-Rotn rad	Y-Rotn rad	Z-Rotn rad
1	0.0000	0.0000	0.0000	0.00000	0.00000	-0.00325
2	0.0000	-0.6788	0.0000	0.00000	0.00000	-0.00051
3	0.0000	0.0000	0.0000	0.00000	0.00000	0.00344

MEMBER FORCES

CASE 1: EXAMPLE APPLIED LOAD

Member	Node	Axial kip	Shear-y kip	Shear-z kip	Torque in.kip	Moment-y in.kip	Moment-z in.kip
1	1	0.000	-12.356	0.000	0.000	0.000	0.000
	2	0.000	1.684	0.000	0.000	0.000	2504.352
2	2	0.000	-5.103	0.000	0.000	0.000	2504.352
	3	0.000	19.017	0.000	0.000	0.000	0.000

Positive Forces (Member Axes):
Axial - Tension Shear - End A sagging
Torque - Right-hand twist Moment - Sagging

SUPPORT REACTIONS

CASE 1: EXAMPLE APPLIED LOAD

Node	Force-X kip	Force-Y kip	Force-Z kip	Moment-X in.kip	Moment-Y in.kip	Moment-Z in.kip
1	0.000	12.356	0.000	0.000	0.000	0.000
2	0.000	6.788	0.000	0.000	0.000	0.000
3	0.000	19.017	0.000	0.000	0.000	0.000

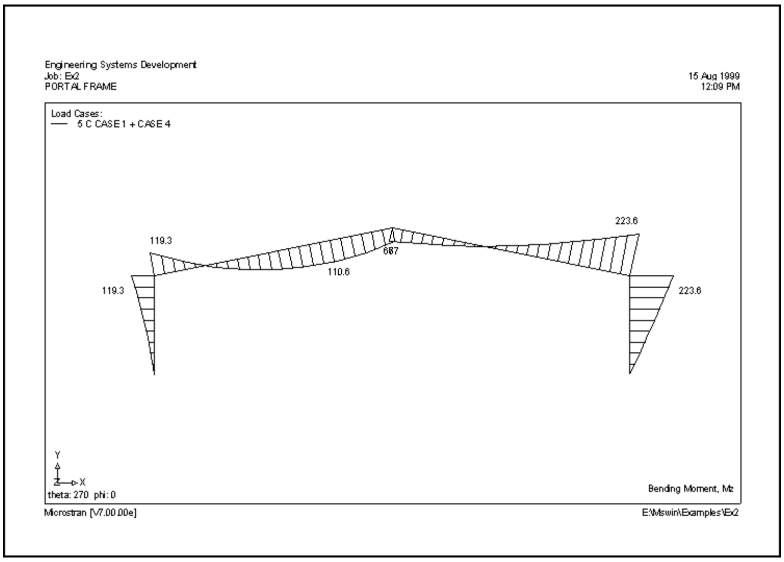
SUM: 0.000 38.160 0.000 (all nodes)

Max. residual: -4.547E-13 at DOFN: 6

(Reactions act on structure in positive global axis directions.)

Example 2 – Portal Frame

Example 2 is a portal frame in the XY plane. The structure and bending moment diagram are shown in the accompanying output. The columns are pinned at the base nodes by allowing Z rotations in the restraint codes for the support nodes, 1 and 5.



EXAMPLE 2 – PORTAL FRAME

Pinned bases could also have been modelled by making the support nodes fully fixed (restraint code 111111) and setting the *z* moment releases at the lower ends of members 1 and 4 (the release codes would then have been 000001 000000 and 000000 000001 respectively). Note that if this were done, no rotational displacement would be computed at the base of each column.

The report shows forces and displacements at 1/5th points along each member for load case 7. Note that you may obtain results along the length of the members without having to use additional nodes. Up to 40 segments per member may be selected for the tabulation of forces and displacements along members.

Archive File

```
* PORTAL FRAME
*
*
*
VERS    4
TYPE    3
VERT    2
UNIT    1 m   kN  t   C

NODE    1      .0000      .0000      .0000 111110
NODE    2      .0000     5.0000      .0000 001110
NODE    3    12.0000     7.4000      .0000 001110
NODE    4    24.0000     5.0000      .0000 001110
NODE    5    24.0000      .0000      .0000 111110

MEMB    1     1     2     D     1     1    000000 000000
MEMB    2     2     3     D     2     1    000000 000000
MEMB    3     3     4     D     2     1    000000 000000
MEMB    4     4     5     D     1     1    000000 000000

PROP    1 LIBR Asw.lib   610UB125        Y
      1.6000E-02  .0000      .0000     1.5600E-06 3.9300E-05 9.8600E-04
PROP    2 LIBR Asw.lib   530UB92.4       Y
      1.1800E-02  .0000      .0000     7.7500E-07 2.3800E-05 5.5400E-04

MATL    1 2.000E+08 3.000E-01 7.850E+00 1.080E-05

CASE    1 DEAD LOAD
MBLD    2  UNIF  FY  GL           -4.000
MBLD    3  UNIF  FY  GL           -4.000

CASE    2 CONCENTRATED LOAD AT APEX
NDLD    3       .000    -50.000       .000       .000       .000       .000

CASE    3 LATERAL LOAD
NDLD    2     50.000       .000       .000       .000       .000       .000

CASE    4 WIND LOAD
MBLD    1  UNIF  FX  GL            3.500
MBLD    2  UNIF  FY  LO           -2.500
MBLD    3  UNIF  FY  LO            1.500
MBLD    4  UNIF  FX  GL            1.000

CASE    5 CASE 1 + CASE 4
COMB    1      1.000
COMB    4      1.000

CASE    6 CASE 2 + CASE 3
COMB    2      1.000
COMB    3      1.000

CASE    7 CASE 5 + CASE 6
COMB    5      1.000
COMB    6      1.000

END
```

```
INPUT/ANALYSIS REPORT

Job:    Ex2

Title:  PORTAL FRAME

Type:   Plane frame
Date:   15 Aug 1999
Time:   12:16 PM

Nodes ............................      5
Members ..........................      4
Spring supports ..................      0
Sections .........................      2
Materials ........................      1
Primary load cases ...............      4
Combination load cases ...........      3

Analysis:  Linear elastic

LOAD CASES

  Case Type Analysis Title
     7   C      L    CASE 5 + CASE 6

Analysis Types:
  S - Skipped (not analysed)
  L - Linear
  N - Non-linear

 NODE TABLE NOT PRINTED

 MEMBER TABLE NOT PRINTED

 SECTION PROPERTY TABLE NOT PRINTED

 MATERIAL TABLE NOT PRINTED

CONDITION NUMBER

Maximum condition number: 1.057E+01 at node: 4 DOFN: 1

SIGN CONVENTION

Positive Forces (Member Axes):
  Axial  - Tension              Shear  - End A sagging
  Torque - Right-hand twist     Moment - Sagging
Deflections:
  Global deflections are absolute.
  Local deflections are relative to chord joining displaced end nodes.

MEMBER FORCES AND DEFLECTIONS

CASE 7: CASE 5 + CASE 6

MEMBER 1: Nodes 1 - 2  Section 1: 610UB125 Y

Point Offset    Axial  Shear-y  Shear-z   Torque Moment-y Moment-z
           m       kN       kN       kN      kNm      kNm      kNm
   1    0.00  -76.710    9.458    0.000    0.000    0.000    0.000
   2    1.00  -76.710   12.958    0.000    0.000    0.000  -11.208
   3    2.00  -76.710   16.458    0.000    0.000    0.000  -25.916
   4    3.00  -76.710   19.958    0.000    0.000    0.000  -44.124
   5    4.00  -76.710   23.458    0.000    0.000    0.000  -65.833
   6    5.00  -76.710   26.958    0.000    0.000    0.000  -91.041

Point Offset   X-glob   Y-glob   Z-glob  y-local  z-local
           m        m        m        m        m        m
   1    0.00   0.0000   0.0000   0.0000  0.00000  0.00000
   2    1.00   0.0073   0.0000   0.0000  0.00028  0.00000
   3    2.00   0.0146   0.0000   0.0000  0.00051  0.00000
   4    3.00   0.0221  -0.0001   0.0000  0.00060  0.00000
```

5	4.00	0.0298	-0.0001	0.0000	0.00047	0.00000
6	5.00	0.0378	-0.0001	0.0000	0.00000	0.00000

MEMBER 2: Nodes 2 - 3 Section 2: 530UB92.4 Y

Point	Offset m	Axial kN	Shear-y kN	Shear-z kN	Torque kNm	Moment-y kNm	Moment-z kNm
1	0.00	-90.508	-60.128	0.000	0.000	0.000	-91.041
2	2.45	-88.588	-44.409	0.000	0.000	0.000	36.888
3	4.90	-86.668	-28.690	0.000	0.000	0.000	126.344
4	7.34	-84.748	-12.971	0.000	0.000	0.000	177.328
5	9.79	-82.828	2.748	0.000	0.000	0.000	189.839
6	12.24	-80.908	18.466	0.000	0.000	0.000	163.878

Point	Offset m	X-glob m	Y-glob m	Z-glob m	y-local m	z-local m
1	0.00	0.0378	-0.0001	0.0000	0.00000	0.00000
2	2.45	0.0419	-0.0211	0.0000	-0.01123	0.00000
3	4.90	0.0456	-0.0403	0.0000	-0.02065	0.00000
4	7.34	0.0481	-0.0530	0.0000	-0.02341	0.00000
5	9.79	0.0487	-0.0564	0.0000	-0.01675	0.00000
6	12.24	0.0473	-0.0500	0.0000	0.00000	0.00000

MEMBER 3: Nodes 3 - 4 Section 2: 530UB92.4 Y

Point	Offset m	Axial kN	Shear-y kN	Shear-z kN	Torque kNm	Moment-y kNm	Moment-z kNm
1	0.00	-91.592	34.957	0.000	0.000	0.000	163.878
2	2.45	-93.512	40.885	0.000	0.000	0.000	71.066
3	4.90	-95.432	46.814	0.000	0.000	0.000	-36.257
4	7.34	-97.352	52.743	0.000	0.000	0.000	-158.091
5	9.79	-99.272	58.671	0.000	0.000	0.000	-294.436
6	12.24	-101.192	64.600	0.000	0.000	0.000	-445.291

Point	Offset m	X-glob m	Y-glob m	Z-glob m	y-local m	z-local m
1	0.00	0.0473	-0.0500	0.0000	0.00000	0.00000
2	2.45	0.0502	-0.0352	0.0000	0.00484	0.00000
3	4.90	0.0537	-0.0168	0.0000	0.01345	0.00000
4	7.34	0.0569	-0.0004	0.0000	0.02004	0.00000
5	9.79	0.0584	0.0076	0.0000	0.01801	0.00000
6	12.24	0.0568	-0.0001	0.0000	0.00000	0.00000

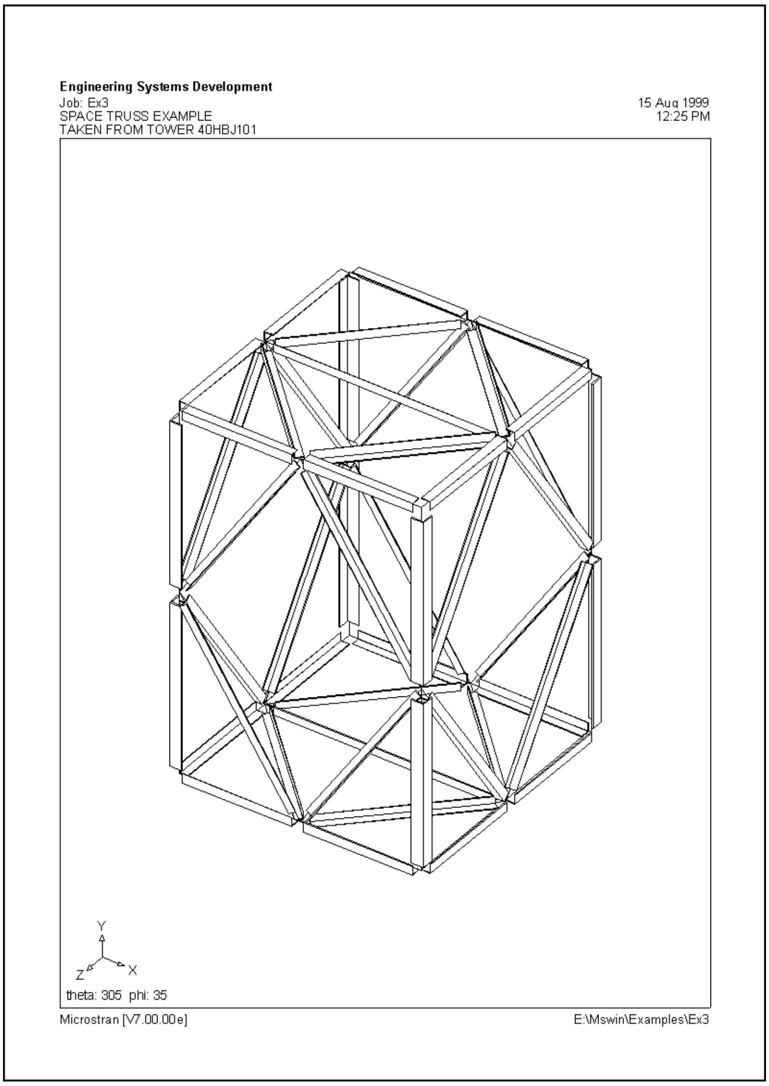
MEMBER 4: Nodes 4 - 5 Section 1: 610UB125 Y

Point	Offset m	Axial kN	Shear-y kN	Shear-z kN	Torque kNm	Moment-y kNm	Moment-z kNm
1	0.00	-83.191	-86.558	0.000	0.000	0.000	-445.291
2	1.00	-83.191	-87.558	0.000	0.000	0.000	-358.233
3	2.00	-83.191	-88.558	0.000	0.000	0.000	-270.174
4	3.00	-83.191	-89.558	0.000	0.000	0.000	-181.116
5	4.00	-83.191	-90.558	0.000	0.000	0.000	-91.058
6	5.00	-83.191	-91.558	0.000	0.000	0.000	0.000

Point	Offset m	X-glob m	Y-glob m	Z-glob m	y-local m	z-local m
1	0.00	0.0568	-0.0001	0.0000	0.00000	0.00000
2	1.00	0.0481	-0.0001	0.0000	0.00273	0.00000
3	2.00	0.0377	-0.0001	0.0000	0.00365	0.00000
4	3.00	0.0259	-0.0001	0.0000	0.00320	0.00000
5	4.00	0.0132	0.0000	0.0000	0.00183	0.00000
6	5.00	0.0000	0.0000	0.0000	0.00000	0.00000

Example 3 – Space Truss

Example 3 is the top section of a high communications tower. It is modelled as a space truss (i.e. all rotational DOF deleted).



EXAMPLE 3 – SPACE TRUSS

A common problem with this type of structure is the presence of coplanar nodes. Coplanar nodes have no out-of-plane stiffness because all attached members lie in a plane. Coplanar nodes will give rise to zero stiffness errors during the analysis or a large condition number, with extremely large displacements for the unrestrained DOF. In this example, a number of "redundant" members have been added to the

structure to provide out-of-plane support to nodes that would otherwise have been coplanar. A coplanar node may also be removed by deleting the DOF normal to the plane (if the plane is normal to a global axis) or by restraining the node with weak springs.

The structure is shown in the accompanying Microstran graphical output. Note in the rendered view that the legs of the tower are oriented with the corners of the angles outwards. The section orientation codes can be seen on some MEMB lines of the archive file. These are required only for detailing – the analysis is not affected by the orientation of the section.

The report shows the analysis results (for a range of nodes and members) for Case 3 , the combination load case.

Archive File

```
* SPACE TRUSS EXAMPLE
* TAKEN FROM TOWER 40HBJ101
*
*
VERS     4
TYPE     4
VERT     2
UNIT     1 m   kN  t   C

NODE   261    -1.0000   107.0000     1.0000 111111
NODE   262     1.0000   107.0000     1.0000 111111
NODE   263     1.0000   107.0000    -1.0000 111111
NODE   264    -1.0000   107.0000    -1.0000 111111
NODE   265    -1.0000   108.5000     1.0000 000111
NODE   266     1.0000   108.5000     1.0000 000111
NODE   267     1.0000   108.5000    -1.0000 000111
NODE   268    -1.0000   108.5000    -1.0000 000111
NODE   269    -1.0000   110.0000     1.0000 000111
NODE   270     1.0000   110.0000     1.0000 000111
NODE   271     1.0000   110.0000    -1.0000 000111
NODE   272    -1.0000   110.0000    -1.0000 000111
NODE   701      .0000   107.0000     1.0000 111111
NODE   702     1.0000   107.0000      .0000 111111
NODE   703      .0000   107.0000    -1.0000 111111
NODE   704    -1.0000   107.0000      .0000 111111
NODE   705      .0000   110.0000     1.0000 000111
NODE   706     1.0000   110.0000      .0000 000111
NODE   707      .0000   110.0000    -1.0000 000111
NODE   708    -1.0000   110.0000      .0000 000111

MEMB    66   261   265    X      4     1    000000 000000
MEMB    67   265   269    X      4     1    000000 000000
MEMB   133   262   266    X      4     1    000000 000000 01
MEMB   134   266   270    X      4     1    000000 000000 01
MEMB   200   263   267    X      4     1    000000 000000 11
MEMB   201   267   271    X      4     1    000000 000000 11
MEMB   267   264   268    X      4     1    000000 000000 10
MEMB   268   268   272    X      4     1    000000 000000 10
MEMB   578   261   701    Y      7     1    000000 000000 10
MEMB   579   701   262    Y      7     1    000000 000000 10
MEMB   580   701   265    Y      8     1    000000 000000
MEMB   581   701   266    Y      8     1    000000 000000
MEMB   582   265   705    Y      8     1    000000 000000
MEMB   583   705   266    Y      8     1    000000 000000
MEMB   584   269   705    Y      7     1    000000 000000 11
MEMB   585   705   270    Y      7     1    000000 000000 11
MEMB   608   262   702    Y      7     1    000000 000000 10
MEMB   609   702   263    Y      7     1    000000 000000 10
MEMB   610   702   266    Y      8     1    000000 000000
MEMB   611   702   267    Y      8     1    000000 000000
MEMB   612   266   706    Y      8     1    000000 000000
MEMB   613   267   706    Y      8     1    000000 000000
MEMB   614   270   706    Y      7     1    000000 000000 11
MEMB   615   706   271    Y      7     1    000000 000000 11
MEMB   638   263   703    Y      7     1    000000 000000 10
MEMB   639   703   264    Y      7     1    000000 000000 10
MEMB   640   703   267    Y      8     1    000000 000000
MEMB   641   703   268    Y      8     1    000000 000000
MEMB   642   267   707    Y      8     1    000000 000000
MEMB   643   268   707    Y      8     1    000000 000000
MEMB   644   271   707    Y      7     1    000000 000000 11
MEMB   645   707   272    Y      7     1    000000 000000 11
```

```
MEMB   668   264   704    Y      7     1    000000 000000 10
MEMB   669   704   261    Y      7     1    000000 000000 10
MEMB   670   704   268    Y      8     1    000000 000000
MEMB   671   704   265    Y      8     1    000000 000000
MEMB   672   268   708    Y      8     1    000000 000000
MEMB   673   265   708    Y      8     1    000000 000000
MEMB   674   272   708    Y      7     1    000000 000000 11
MEMB   675   708   269    Y      7     1    000000 000000 11
MEMB   798   705   708    Y      7     1    000000 000000
MEMB   799   705   706    Y      7     1    000000 000000
MEMB   800   707   708    Y      7     1    000000 000000
MEMB   801   707   706    Y      7     1    000000 000000
MEMB   976   701   702    Y      8     1    000000 000000
MEMB   977   702   703    Y      8     1    000000 000000
MEMB   978   703   704    Y      8     1    000000 000000
MEMB   979   704   701    Y      8     1    000000 000000
MEMB   980   702   704    Y      8     1    000000 000000
MEMB   981   706   708    Y      8     1    000000 000000

PROP     4 LIBR Asw       100X100X10EA    Y
      1.8100E-03  .0000      .0000     5.6200E-08 1.7000E-06 1.7000E-06
PROP     7 LIBR Asw       75X75X6EA       Y
      8.6700E-04  .0000      .0000     1.1200E-08 4.5500E-07 4.5500E-07
PROP     8 LIBR Asw       65X65X6EA       Y
      7.4800E-04  .0000      .0000     9.3700E-09 2.9600E-07 2.9600E-07

MATL     1 2.000E+08 3.000E-01 7.850E+00 1.080E-05

CASE     1 WIND AND DEAD LOADS
NDLD   261      4.450     -1.388       .000       .000       .000       .000
NDLD   262       .000     -1.388       .000       .000       .000       .000
NDLD   263       .000     -1.388       .000       .000       .000       .000
NDLD   264      4.450     -1.388       .000       .000       .000       .000
NDLD   269      4.450     -3.595       .000       .000       .000       .000
NDLD   270       .000     -3.595       .000       .000       .000       .000
NDLD   271       .000     -3.595       .000       .000       .000       .000
NDLD   272      4.450     -3.595       .000       .000       .000       .000

CASE     2 ANTENNA LOADS
NDLD   270      7.440     -1.079       .000       .000       .000       .000
NDLD   272      7.440     -1.079       .000       .000       .000       .000

CASE     3 CASE 1 + CASE 2
COMB     1      1.000
COMB     2      1.000

END
```

INPUT/ANALYSIS REPORT

```
Job:    Ex3

Title:  SPACE TRUSS EXAMPLE
        TAKEN FROM TOWER 40HBJ101
Type:   Space truss
Date:   15 Aug 1999
Time:    1:00 PM

Nodes .............................    20
Members ...........................    50
Spring supports ...................     0
Sections ..........................     3
Materials .........................     1
Primary load cases ................     2
Combination load cases ............     1

Analysis:  Linear elastic
```

LOAD CASES

```
  Case Type Analysis Title
     3   C      L    CASE 1 + CASE 2

Analysis Types:
  S - Skipped (not analysed)
  L - Linear
  N - Non-linear

 NODE TABLE NOT PRINTED

 MEMBER TABLE NOT PRINTED

 SECTION PROPERTY TABLE NOT PRINTED

 MATERIAL TABLE NOT PRINTED
```

CONDITION NUMBER

Maximum condition number: 7.878E+00 at node: 271 DOFN: 3

NODE DISPLACEMENTS

CASE 3: CASE 1 + CASE 2

Node	X-Disp m	Y-Disp m	Z-Disp m	X-Rotn rad	Y-Rotn rad	Z-Rotn rad
261	0.0000	0.0000	0.0000	0.00000	0.00000	0.00000
262	0.0000	0.0000	0.0000	0.00000	0.00000	0.00000
263	0.0000	0.0000	0.0000	0.00000	0.00000	0.00000
264	0.0000	0.0000	0.0000	0.00000	0.00000	0.00000
265	0.0003	0.0001	-0.0001	0.00000	0.00000	0.00000
266	0.0004	-0.0001	0.0001	0.00000	0.00000	0.00000
267	0.0004	-0.0001	-0.0001	0.00000	0.00000	0.00000
268	0.0003	0.0001	0.0001	0.00000	0.00000	0.00000
269	0.0007	0.0000	0.0000	0.00000	0.00000	0.00000
270	0.0007	-0.0001	0.0000	0.00000	0.00000	0.00000
271	0.0007	-0.0001	0.0000	0.00000	0.00000	0.00000
272	0.0008	0.0000	0.0000	0.00000	0.00000	0.00000
701	0.0000	0.0000	0.0000	0.00000	0.00000	0.00000
702	0.0000	0.0000	0.0000	0.00000	0.00000	0.00000
703	0.0000	0.0000	0.0000	0.00000	0.00000	0.00000
704	0.0000	0.0000	0.0000	0.00000	0.00000	0.00000
705	0.0007	0.0000	0.0000	0.00000	0.00000	0.00000
706	0.0007	-0.0002	0.0000	0.00000	0.00000	0.00000
707	0.0007	0.0000	0.0000	0.00000	0.00000	0.00000
708	0.0007	0.0001	0.0000	0.00000	0.00000	0.00000

MEMBER FORCES

CASE 3: CASE 1 + CASE 2

Member	Node	Axial kN	Shear-y kN	Shear-z kN	Torque kNm	Moment-y kNm	Moment-z kNm
670	704	-0.075	0.000	0.000	0.000	0.000	0.000
	268	-0.075	0.000	0.000	0.000	0.000	0.000
671	704	0.075	0.000	0.000	0.000	0.000	0.000
	265	0.075	0.000	0.000	0.000	0.000	0.000
672	268	0.075	0.000	0.000	0.000	0.000	0.000
	708	0.075	0.000	0.000	0.000	0.000	0.000
673	265	-0.075	0.000	0.000	0.000	0.000	0.000
	708	-0.075	0.000	0.000	0.000	0.000	0.000
674	272	0.000	0.000	0.000	0.000	0.000	0.000
	708	0.000	0.000	0.000	0.000	0.000	0.000
675	708	0.000	0.000	0.000	0.000	0.000	0.000
	269	0.000	0.000	0.000	0.000	0.000	0.000
798	705	0.059	0.000	0.000	0.000	0.000	0.000
	708	0.059	0.000	0.000	0.000	0.000	0.000
799	705	-0.059	0.000	0.000	0.000	0.000	0.000
	706	-0.059	0.000	0.000	0.000	0.000	0.000
800	707	-0.059	0.000	0.000	0.000	0.000	0.000
	708	-0.059	0.000	0.000	0.000	0.000	0.000
801	707	0.059	0.000	0.000	0.000	0.000	0.000
	706	0.059	0.000	0.000	0.000	0.000	0.000
976	701	0.000	0.000	0.000	0.000	0.000	0.000
	702	0.000	0.000	0.000	0.000	0.000	0.000
977	702	0.000	0.000	0.000	0.000	0.000	0.000
	703	0.000	0.000	0.000	0.000	0.000	0.000
978	703	0.000	0.000	0.000	0.000	0.000	0.000
	704	0.000	0.000	0.000	0.000	0.000	0.000
979	704	0.000	0.000	0.000	0.000	0.000	0.000
	701	0.000	0.000	0.000	0.000	0.000	0.000
980	702	0.000	0.000	0.000	0.000	0.000	0.000
	704	0.000	0.000	0.000	0.000	0.000	0.000
981	706	0.000	0.000	0.000	0.000	0.000	0.000
	708	0.000	0.000	0.000	0.000	0.000	0.000

Positive Forces (Member Axes):
Axial - Tension Shear - End A sagging
Torque - Right-hand twist Moment - Sagging

SUPPORT REACTIONS

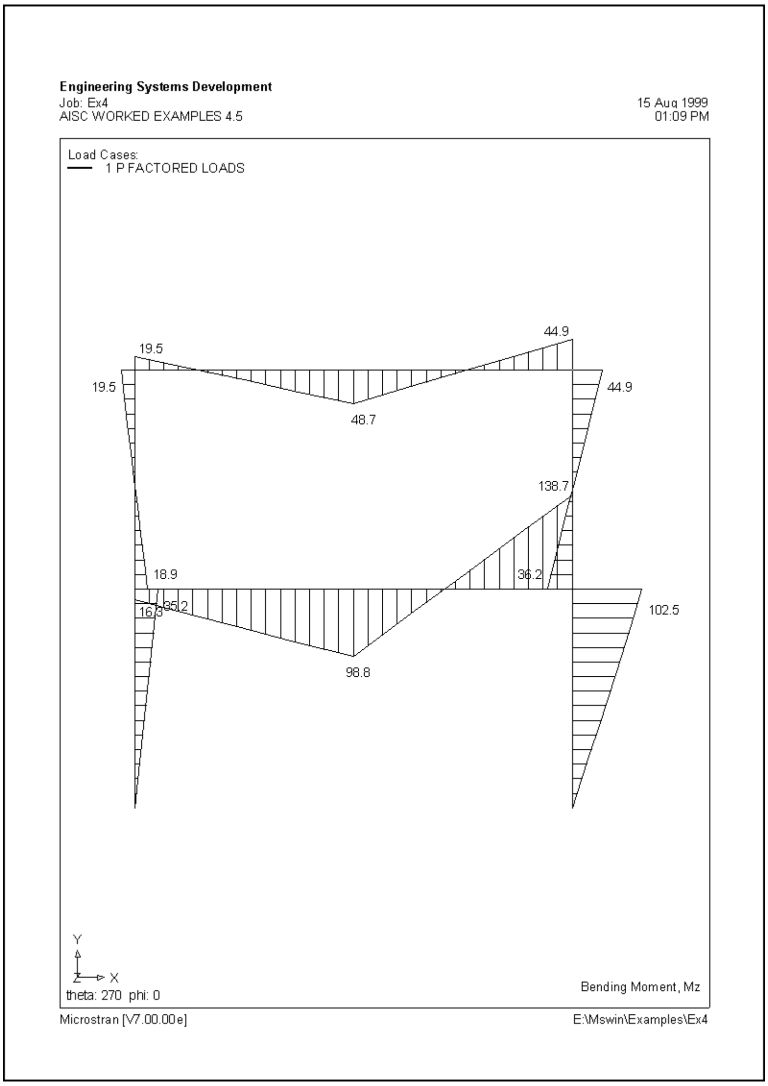
CASE 3: CASE 1 + CASE 2

Node	Force-X kN	Force-Y kN	Force-Z kN	Moment-X kNm	Moment-Y kNm	Moment-Z kNm
261	-4.450	-12.602	0.000	0.000	0.000	0.000
262	0.000	23.647	0.000	0.000	0.000	0.000
263	0.000	23.068	0.000	0.000	0.000	0.000
264	-4.450	-12.023	0.000	0.000	0.000	0.000
701	-11.807	0.000	0.000	0.000	0.000	0.000
702	0.000	0.000	0.083	0.000	0.000	0.000
703	-11.973	0.000	0.000	0.000	0.000	0.000
704	0.000	0.000	-0.083	0.000	0.000	0.000
SUM:	-32.680	22.090	0.000	(all nodes)		

(Reactions act on structure in positive global axis directions.)

Example 4 – Non-Linear Analysis

Example 4 demonstrates a non-linear analysis of a two-storey plane frame. With node coordinate update and axial force effects both in effect, the analysis requires 4 iterations. Essentially identical results are obtained with 2 iterations if the node coordinate update check box is cleared. The report shows the analysis results. Note that the settings of the analysis flags are shown on the report.



EXAMPLE 4 – NON-LINEAR ANALYSIS

Archive File

```
* AISC WORKED EXAMPLES 4.5
*
*
*
VERS     4
TYPE     3
VERT     2
UNIT     1 m   kN  t   C

NODE     1       .0000       .0000       .0000 111110
NODE     2    10.0000       .0000       .0000 111110
NODE    11       .0000      5.0000       .0000 001110
NODE    12    10.0000      5.0000       .0000 001110
NODE    21       .0000     10.0000       .0000 001110
NODE    22    10.0000     10.0000       .0000 001110

MEMB     1    11    12     D    10     1    000000 000000
MEMB    11    21    22     D    11     1    000000 000000
MEMB    21     1    11     D    21     1    000000 000000
MEMB    22    11    21     D    22     1    000000 000000
MEMB    31     2    12     D    21     1    000000 000000
MEMB    32    12    22     D    22     1    000000 000000

PROP    10 LIBR Asw.lib   310UB40.4       Y
      5.2100E-03  .0000      .0000     1.5700E-07 7.6500E-06 8.6400E-05
PROP    11 LIBR Asw.lib   200UB25.4       Y
      3.2300E-03  .0000      .0000     6.2700E-08 3.0600E-06 2.3600E-05
PROP    21 LIBR Asw.lib   200UC52.2       Y
      6.6600E-03  .0000      .0000     3.2500E-07 1.7700E-05 5.2800E-05
PROP    22 LIBR Asw.lib   150UC30.0       Y
      3.8600E-03  .0000      .0000     1.0900E-07 5.6200E-06 1.7600E-05

MATL     1 2.000E+08 2.000E-01 7.850E+00 1.080E-05

CASE     1 FACTORED LOADS
NDLD    11      16.000    -32.000       .000       .000       .000       .000
NDLD    12        .000    -32.000       .000       .000       .000       .000
NDLD    21       8.000    -16.000       .000       .000       .000       .000
NDLD    22        .000    -16.000       .000       .000       .000       .000
MBLD     1  CONC  FY  GL  FR      -64.000       .500
MBLD    11  CONC  FY  GL  FR      -32.000       .500

END
```

INPUT/ANALYSIS REPORT

```
Job:    Ex4

Title:  AISC WORKED EXAMPLES 4.5

Type:   Plane frame
Date:   15 Aug 1999
Time:    1:38 PM

Nodes ............................      6
Members ..........................      6
Spring supports ..................      0
Sections .........................      4
Materials ........................      1
Primary load cases ...............      1
Combination load cases ...........      0

Analysis:  Non-linear elastic
Update node coordinates ..........      Y
Small displacement theory ........      Y
Include axial force effects ......      Y
Include flexural shortening ......      N
Convergence criterion:  Residual
Convergence tolerance .........  5.000E-04
```

LOAD CASES

```
  Case Type Analysis Title
     1   P      N    FACTORED LOADS

Analysis Types:
  S - Skipped (not analysed)
  L - Linear
  N - Non-linear

 NODE TABLE NOT PRINTED

 MEMBER TABLE NOT PRINTED

 SECTION PROPERTY TABLE NOT PRINTED

 MATERIAL TABLE NOT PRINTED
```

APPLIED LOADING

CASE   1: FACTORED LOADS

Node Loads

Node	X Force kN	Y Force kN	Z Force kN	X Moment kNm	Y Moment kNm	Z Moment kNm
11	16.000	-32.000	0.000	0.000	0.000	0.000
12	0.000	-32.000	0.000	0.000	0.000	0.000
21	8.000	-16.000	0.000	0.000	0.000	0.000
22	0.000	-16.000	0.000	0.000	0.000	0.000

Member Loads

Member	Form	T	A	S	F1	X1	F2	X2
1	CONC	FY	GL	FR	-64.000	0.500		
11	CONC	FY	GL	FR	-32.000	0.500		

Sum of Applied Loads (Global Axes):

```
  FX:      24.000    FY:    -192.000    FZ:       0.000
```

NODE DISPLACEMENTS

CASE 1: FACTORED LOADS

Node	X-Disp m	Y-Disp m	Z-Disp m	X-Rotn rad	Y-Rotn rad	Z-Rotn rad
1	0.0000	0.0000	0.0000	0.00000	0.00000	-0.02137
2	0.0000	0.0000	0.0000	0.00000	0.00000	-0.02689
11	0.0927	-0.0012	0.0000	0.00000	0.00000	-0.01292
12	0.0927	-0.0013	0.0000	0.00000	0.00000	-0.00206
21	0.1356	-0.0015	0.0000	0.00000	0.00000	-0.01334
22	0.1353	-0.0017	0.0000	0.00000	0.00000	0.00428

MEMBER FORCES

CASE 1: FACTORED LOADS

Member	Node	Axial kN	Shear-y kN	Shear-z kN	Torque kNm	Moment-y kNm	Moment-z kNm
1	11	-2.479	-16.501	0.000	0.000	0.000	16.251
	12	-2.480	47.499	0.000	0.000	0.000	-138.736
11	21	-15.929	-13.456	0.000	0.000	0.000	-19.481
	22	-15.929	18.544	0.000	0.000	0.000	-44.921
21	1	-77.837	-7.031	0.000	0.000	0.000	0.000
	11	-77.837	-7.031	0.000	0.000	0.000	35.151
22	11	-29.522	7.677	0.000	0.000	0.000	18.901
	21	-29.522	7.677	0.000	0.000	0.000	-19.481
31	2	-113.681	-20.511	0.000	0.000	0.000	-0.002
	12	-113.681	-20.511	0.000	0.000	0.000	102.545
32	12	-34.407	-16.223	0.000	0.000	0.000	-36.188
	22	-34.407	-16.223	0.000	0.000	0.000	44.921

Positive Forces (Member Axes):
Axial - Tension Shear - End A sagging
Torque - Right-hand twist Moment - Sagging

SUPPORT REACTIONS

CASE 1: FACTORED LOADS

Node	Force-X kN	Force-Y kN	Force-Z kN	Moment-X kNm	Moment-Y kNm	Moment-Z kNm
1	-5.586	77.954	0.000	0.000	0.000	0.000
2	-18.401	114.041	0.000	0.000	0.000	0.000

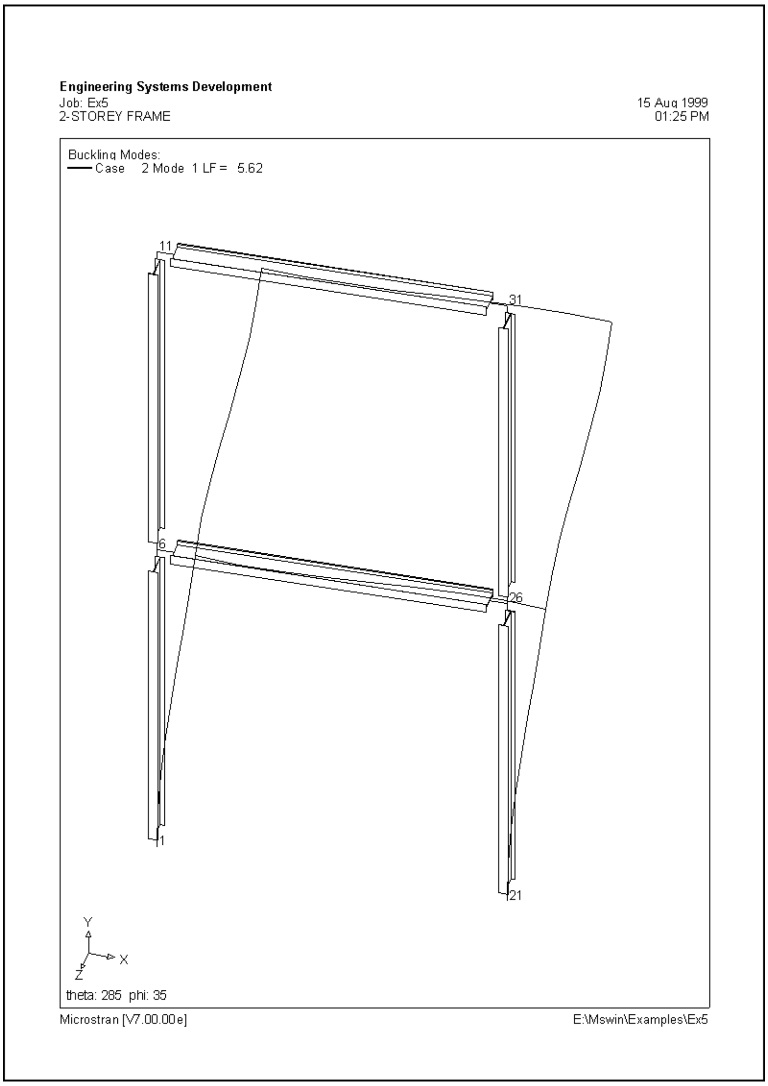
SUM: -23.987 191.996 0.000 (all nodes)

Max. residual: 7.208E-03 at DOFN: 1 1

(Reactions act on structure in positive global axis directions.)

Example 5 – Elastic Critical Load Analysis

Example 5 demonstrates an elastic critical load analysis of a two-storey plane frame. The accompanying diagram shows the first buckling mode shape superimposed on a rendered view of the structure. The report shows the structure geometry and the table of effective lengths.



EXAMPLE 5 ELASTIC CRITICAL LOAD ANALYSIS

Archive File

```
* 2-STOREY FRAME
*
*
*
VERS     4
TYPE     5
VERT     2
UNIT     1 m   kN  t   C

NODE     1       .0000       .0000       .0000 111111
NODE     6       .0000     10.0000       .0000 001110
NODE    11       .0000     20.0000       .0000 001110
NODE    21     10.0000       .0000       .0000 111111
NODE    26     10.0000     10.0000       .0000 001110
NODE    31     10.0000     20.0000       .0000 001110

MEMB     1     1     6     D     1     1    000000 000000
MEMB     2     6    11     D     1     1    000000 000000
MEMB     3    21    26     D     1     1    000000 000000
MEMB     4    26    31     D     1     1    000000 000000
MEMB     5     6    26     D     2     1    000000 000000
MEMB     6    11    31     D     2     1    000000 000000

PROP     1 LIBR Asw.lib   760UB147        X
      1.8800E-02  .0000      .0000     1.6100E-06 1.6900E-03 5.4700E-05
PROP     2 LIBR Asw.lib   760UB147        X
      1.8800E-02  .0000      .0000     1.6100E-06 1.6900E-03 5.4700E-05

MATL     1 2.000E+08 2.000E-01 7.850E+00 0.000E+00

CASE     2 VERTICAL AND HORIZONTAL LOADING
NDLD    11     10.000   -100.000       .000       .000       .000       .000
NDLD    31       .000   -100.000       .000       .000       .000       .000

END
```

INPUT/ANALYSIS REPORT

```
Job:    Ex5

Title:  2-STOREY FRAME

Type:   Space frame
Date:   15 Aug 1999
Time:    1:29 PM

Nodes ............................      6
Members ..........................      6
Spring supports ..................      0
Sections .........................      2
Materials ........................      1
Primary load cases ...............      1
Combination load cases ...........      0
```

LOAD CASES

```
  Case Type Analysis Title
     2   P      S    VERTICAL AND HORIZONTAL LOADING

Analysis Types:
  S - Skipped (not analysed)
  L - Linear
  N - Non-linear
```

NODE COORDINATES

Node	X m	Y m	Z m	Restraint
1	0.000	0.000	0.000	111111
6	0.000	10.000	0.000	001110
11	0.000	20.000	0.000	001110
21	10.000	0.000	0.000	111111
26	10.000	10.000	0.000	001110
31	10.000	20.000	0.000	001110

MEMBER DEFINITION

Member	A	B	C	Prop	Matl	Rel-A	Rel-B	Length m
1	1	6	-X	1	1	000000	000000	10.000
2	6	11	-X	1	1	000000	000000	10.000
3	21	26	-X	1	1	000000	000000	10.000
4	26	31	-X	1	1	000000	000000	10.000
5	6	26	Y	2	1	000000	000000	10.000
6	11	31	Y	2	1	000000	000000	10.000

LIBRARY SECTIONS

Section	Library	Name	Axis	Comment
1	Asw	760UB147	X	
2	Asw	760UB147	X	

SECTION PROPERTIES

Section	Ax m2	Ay m2	Az m2	J m4	Iy m4	Iz m4	fact
1	1.880E-02	0.000E+00	0.000E+00	1.610E-06	1.690E-03	5.470E-05	
2	1.880E-02	0.000E+00	0.000E+00	1.610E-06	1.690E-03	5.470E-05	

MATERIAL PROPERTIES

Material	E	u	Density	Alpha
	kN/m2		t/m3	/deg C
1	2.000E+08	0.2000	7.850E+00	0.000E+00

TABLE OF QUANTITIES

MATERIAL 1

Section	Name	Length	Mass	Comment
		m	tonne	
1	760UB147	40.000	5.903	
2	760UB147	20.000	2.952	
		----------	----------	
		60.000	8.855	

APPLIED LOADING

CASE 2: VERTICAL AND HORIZONTAL LOADING

Node Loads

Node	X Force	Y Force	Z Force	X Moment	Y Moment	Z Moment
	kN	kN	kN	kNm	kNm	kNm
11	10.000	-100.000	0.000	0.000	0.000	0.000
31	0.000	-100.000	0.000	0.000	0.000	0.000

EFFECTIVE LENGTH FACTORS

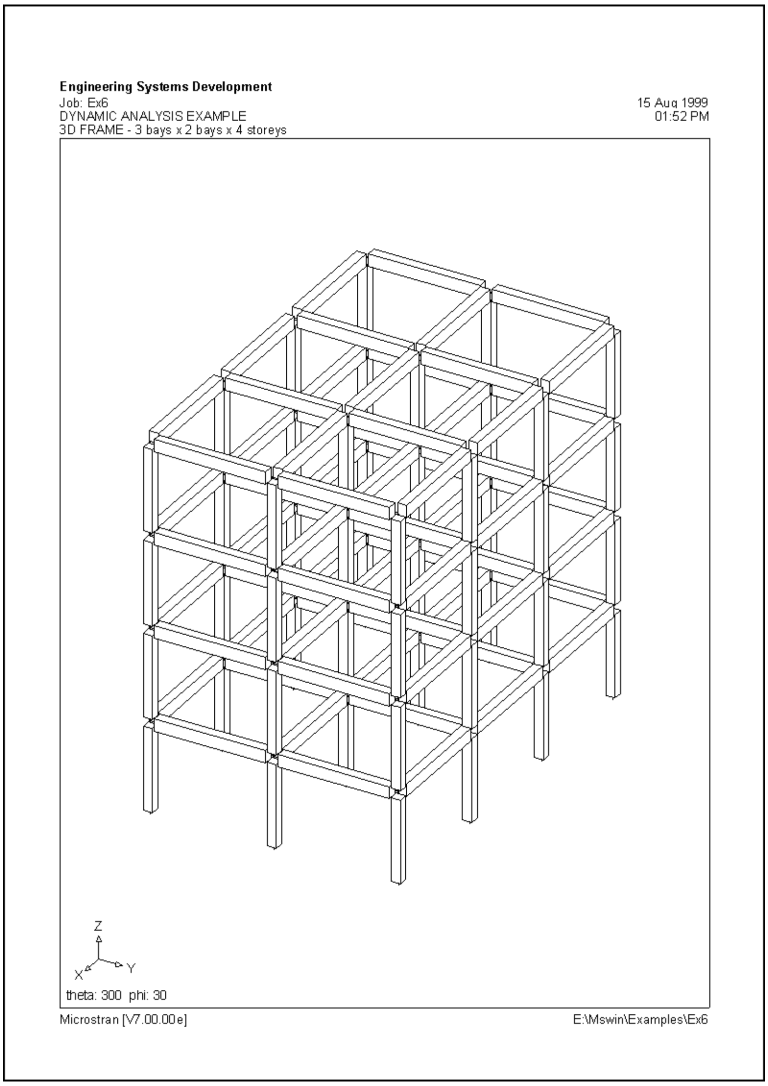
CASE 2: VERTICAL AND HORIZONTAL LOADING

Elastic Critical Load Factor: 5.62

Member	Length	Pcrit	Pey	ky	Pez	kz
	m	kN	kN		kN	
1	10.000	-485.13	33359.25	8.29	1079.73	1.49
2	10.000	-531.08	33359.25	7.93	1079.73	1.43
3	10.000	-638.31	33359.25	7.23	1079.73	1.30
4	10.000	-592.35	33359.25	7.50	1079.73	1.35
5	10.000	0.00	33359.25	3451.13	1079.73	620.89
6	10.000	-28.08	33359.25	34.46	1079.73	6.20

Example 6 – Dynamic Analysis

Example 6 demonstrates a dynamic analysis of a four-storey reinforced concrete building frame. The accompanying graphical output shows a rendered view of the structure and the third mode shape superimposed on a diagram of the structure. The output report shows the node masses and the frequency and mode shape for the first mode.



EXAMPLE 6 – DYNAMIC ANALYSIS

Archive File

```
* DYNAMIC ANALYSIS EXAMPLE
* 3D FRAME - 3 bays x 2 bays x 4 storeys
*
*

VERS     4
TYPE     5
VERT     3
UNIT     1 m   kN  T   C

NODE     1     0.0000     0.0000     0.0000 111111
NODE     2     4.0000     0.0000     0.0000 111111
NODE     3     8.0000     0.0000     0.0000 111111
NODE     4    12.0000     0.0000     0.0000 111111
NODE    11     0.0000     4.0000     0.0000 111111
NODE    12     4.0000     4.0000     0.0000 111111
NODE    13     8.0000     4.0000     0.0000 111111
NODE    14    12.0000     4.0000     0.0000 111111
NODE    21     0.0000     8.0000     0.0000 111111
NODE    22     4.0000     8.0000     0.0000 111111
NODE    23     8.0000     8.0000     0.0000 111111
NODE    24    12.0000     8.0000     0.0000 111111
NODE    31     0.0000     0.0000     3.0000 000000
NODE    32     4.0000     0.0000     3.0000 000000
NODE    33     8.0000     0.0000     3.0000 000000
NODE    34    12.0000     0.0000     3.0000 000000
NODE    41     0.0000     4.0000     3.0000 000000
NODE    42     4.0000     4.0000     3.0000 000000
NODE    43     8.0000     4.0000     3.0000 000000
NODE    44    12.0000     4.0000     3.0000 000000
NODE    51     0.0000     8.0000     3.0000 000000
NODE    52     4.0000     8.0000     3.0000 000000
NODE    53     8.0000     8.0000     3.0000 000000
NODE    54    12.0000     8.0000     3.0000 000000
NODE    61     0.0000     0.0000     6.0000 000000
NODE    62     4.0000     0.0000     6.0000 000000
NODE    63     8.0000     0.0000     6.0000 000000
NODE    64    12.0000     0.0000     6.0000 000000
NODE    71     0.0000     4.0000     6.0000 000000
NODE    72     4.0000     4.0000     6.0000 000000
NODE    73     8.0000     4.0000     6.0000 000000
NODE    74    12.0000     4.0000     6.0000 000000
NODE    81     0.0000     8.0000     6.0000 000000
NODE    82     4.0000     8.0000     6.0000 000000
NODE    83     8.0000     8.0000     6.0000 000000
NODE    84    12.0000     8.0000     6.0000 000000
NODE    91     0.0000     0.0000     9.0000 000000
NODE    92     4.0000     0.0000     9.0000 000000
NODE    93     8.0000     0.0000     9.0000 000000
NODE    94    12.0000     0.0000     9.0000 000000
NODE   101     0.0000     4.0000     9.0000 000000
NODE   102     4.0000     4.0000     9.0000 000000
NODE   103     8.0000     4.0000     9.0000 000000
NODE   104    12.0000     4.0000     9.0000 000000
NODE   111     0.0000     8.0000     9.0000 000000
NODE   112     4.0000     8.0000     9.0000 000000
NODE   113     8.0000     8.0000     9.0000 000000
NODE   114    12.0000     8.0000     9.0000 000000
NODE   121     0.0000     0.0000    12.0000 000000
NODE   122     4.0000     0.0000    12.0000 000000
NODE   123     8.0000     0.0000    12.0000 000000
NODE   124    12.0000     0.0000    12.0000 000000
NODE   131     0.0000     4.0000    12.0000 000000
NODE   132     4.0000     4.0000    12.0000 000000
NODE   133     8.0000     4.0000    12.0000 000000
NODE   134    12.0000     4.0000    12.0000 000000
NODE   141     0.0000     8.0000    12.0000 000000
NODE   142     4.0000     8.0000    12.0000 000000
NODE   143     8.0000     8.0000    12.0000 000000
NODE   144    12.0000     8.0000    12.0000 000000

NMAS    31      10.0000
NMAS    32      10.0000
NMAS    33      10.0000
NMAS    34      10.0000
NMAS    41      10.0000
NMAS    42      10.0000
NMAS    43      10.0000
NMAS    44      10.0000
NMAS    51      10.0000
NMAS    52      10.0000
NMAS    53      10.0000
NMAS    54      10.0000
NMAS    61      10.0000
NMAS    62      10.0000
NMAS    63      10.0000
NMAS    64      10.0000
NMAS    71      10.0000
NMAS    72      10.0000
NMAS    73      10.0000
NMAS    74      10.0000
NMAS    81      10.0000
NMAS    82      10.0000
NMAS    83      10.0000
NMAS    84      10.0000
NMAS    91      10.0000
NMAS    92      10.0000
```

```
NMAS    93        10.0000
NMAS    94        10.0000
NMAS   101        10.0000
NMAS   102        10.0000
NMAS   103        10.0000
NMAS   104        10.0000
NMAS   111        10.0000
NMAS   112        10.0000
NMAS   113        10.0000
NMAS   114        10.0000
NMAS   121        10.0000
NMAS   122        10.0000
NMAS   123        10.0000
NMAS   124        10.0000
NMAS   131        10.0000
NMAS   132        10.0000
NMAS   133        10.0000
NMAS   134        10.0000
NMAS   141        10.0000
NMAS   142        10.0000
NMAS   143        10.0000
NMAS   144        10.0000

MEMB     1    31    32    Z     10     1    000000 000000
MEMB     2    32    33    Z     10     1    000000 000000
MEMB     3    33    34    Z     10     1    000000 000000
MEMB    11    41    42    Z     10     1    000000 000000
MEMB    12    42    43    Z     10     1    000000 000000
MEMB    13    43    44    Z     10     1    000000 000000
MEMB    21    51    52    Z     10     1    000000 000000
MEMB    22    52    53    Z     10     1    000000 000000
MEMB    23    53    54    Z     10     1    000000 000000
MEMB    31    31    41    Z     20     1    000000 000000
MEMB    32    41    51    Z     20     1    000000 000000
MEMB    41    32    42    Z     20     1    000000 000000
MEMB    42    42    52    Z     20     1    000000 000000
MEMB    51    33    43    Z     20     1    000000 000000
MEMB    52    43    53    Z     20     1    000000 000000
MEMB    61    34    44    Z     20     1    000000 000000
MEMB    62    44    54    Z     20     1    000000 000000
MEMB    71    61    62    Z     10     1    000000 000000
MEMB    72    62    63    Z     10     1    000000 000000
MEMB    73    63    64    Z     10     1    000000 000000
MEMB    81    71    72    Z     10     1    000000 000000
MEMB    82    72    73    Z     10     1    000000 000000
MEMB    83    73    74    Z     10     1    000000 000000
MEMB    91    81    82    Z     10     1    000000 000000
MEMB    92    82    83    Z     10     1    000000 000000
MEMB    93    83    84    Z     10     1    000000 000000
MEMB   101    61    71    Z     20     1    000000 000000
MEMB   102    71    81    Z     20     1    000000 000000
MEMB   111    62    72    Z     20     1    000000 000000
MEMB   112    72    82    Z     20     1    000000 000000
MEMB   121    63    73    Z     20     1    000000 000000
MEMB   122    73    83    Z     20     1    000000 000000
MEMB   131    64    74    Z     20     1    000000 000000
MEMB   132    74    84    Z     20     1    000000 000000
MEMB   141    91    92    Z     10     1    000000 000000
MEMB   142    92    93    Z     10     1    000000 000000
MEMB   143    93    94    Z     10     1    000000 000000
MEMB   151   101   102    Z     10     1    000000 000000
MEMB   152   102   103    Z     10     1    000000 000000
MEMB   153   103   104    Z     10     1    000000 000000
MEMB   161   111   112    Z     10     1    000000 000000
MEMB   162   112   113    Z     10     1    000000 000000
MEMB   163   113   114    Z     10     1    000000 000000
MEMB   171    91   101    Z     20     1    000000 000000
MEMB   172   101   111    Z     20     1    000000 000000
MEMB   181    92   102    Z     20     1    000000 000000
MEMB   182   102   112    Z     20     1    000000 000000
MEMB   191    93   103    Z     20     1    000000 000000
MEMB   192   103   113    Z     20     1    000000 000000
MEMB   201    94   104    Z     20     1    000000 000000
MEMB   202   104   114    Z     20     1    000000 000000
MEMB   211   121   122    Z     10     1    000000 000000
MEMB   212   122   123    Z     10     1    000000 000000
MEMB   213   123   124    Z     10     1    000000 000000
MEMB   221   131   132    Z     10     1    000000 000000
MEMB   222   132   133    Z     10     1    000000 000000
MEMB   223   133   134    Z     10     1    000000 000000
MEMB   231   141   142    Z     10     1    000000 000000
MEMB   232   142   143    Z     10     1    000000 000000
MEMB   233   143   144    Z     10     1    000000 000000
MEMB   241   121   131    Z     20     1    000000 000000
MEMB   242   131   141    Z     20     1    000000 000000
MEMB   251   122   132    Z     20     1    000000 000000
MEMB   252   132   142    Z     20     1    000000 000000
MEMB   261   123   133    Z     20     1    000000 000000
MEMB   262   133   143    Z     20     1    000000 000000
MEMB   271   124   134    Z     20     1    000000 000000
MEMB   272   134   144    Z     20     1    000000 000000
MEMB   281     1    31   -X     30     2    000000 000000
MEMB   282    31    61   -X     30     2    000000 000000
MEMB   283    61    91   -X     30     2    000000 000000
MEMB   284    91   121   -X     30     2    000000 000000
MEMB   291     2    32   -X     31     2    000000 000000
MEMB   292    32    62   -X     31     2    000000 000000
MEMB   293    62    92   -X     31     2    000000 000000
MEMB   294    92   122   -X     31     2    000000 000000
MEMB   301     3    33   -X     31     2    000000 000000
```

```
MEMB   302    33    63   -X     31     2    000000 000000
MEMB   303    63    93   -X     31     2    000000 000000
MEMB   304    93   123   -X     31     2    000000 000000
MEMB   311     4    34   -X     30     2    000000 000000
MEMB   312    34    64   -X     30     2    000000 000000
MEMB   313    64    94   -X     30     2    000000 000000
MEMB   314    94   124   -X     30     2    000000 000000
MEMB   321    11    41   -X     30     2    000000 000000
MEMB   322    41    71   -X     30     2    000000 000000
MEMB   323    71   101   -X     30     2    000000 000000
MEMB   324   101   131   -X     30     2    000000 000000
MEMB   331    12    42   -X     31     2    000000 000000
MEMB   332    42    72   -X     31     2    000000 000000
MEMB   333    72   102   -X     31     2    000000 000000
MEMB   334   102   132   -X     31     2    000000 000000
MEMB   341    13    43   -X     31     2    000000 000000
MEMB   342    43    73   -X     31     2    000000 000000
MEMB   343    73   103   -X     31     2    000000 000000
MEMB   344   103   133   -X     31     2    000000 000000
MEMB   351    14    44   -X     30     2    000000 000000
MEMB   352    44    74   -X     30     2    000000 000000
MEMB   353    74   104   -X     30     2    000000 000000
MEMB   354   104   134   -X     30     2    000000 000000
MEMB   361    21    51   -X     30     2    000000 000000
MEMB   362    51    81   -X     30     2    000000 000000
MEMB   363    81   111   -X     30     2    000000 000000
MEMB   364   111   141   -X     30     2    000000 000000
MEMB   371    22    52   -X     31     2    000000 000000
MEMB   372    52    82   -X     31     2    000000 000000
MEMB   373    82   112   -X     31     2    000000 000000
MEMB   374   112   142   -X     31     2    000000 000000
MEMB   381    23    53   -X     31     2    000000 000000
MEMB   382    53    83   -X     31     2    000000 000000
MEMB   383    83   113   -X     31     2    000000 000000
MEMB   384   113   143   -X     31     2    000000 000000
MEMB   391    24    54   -X     30     2    000000 000000
MEMB   392    54    84   -X     30     2    000000 000000
MEMB   393    84   114   -X     30     2    000000 000000
MEMB   394   114   144   -X     30     2    000000 000000

PROP    10 SHAP RECT 400X300         X_beams
      0.400     0.300      1.00
PROP    20 SHAP RECT 400X300         Y_beams
      0.400     0.300      1.00
PROP    30 SHAP RECT 300X300         End_X_columns
      0.300     0.300      1.00
PROP    31 SHAP RECT 300X300         Interior_columns
      0.300     0.300      1.00

MATL     1 3.230E+07 2.000E-01 2.450E+00 1.170E-05
MATL     2 3.610E+07 2.000E-01 2.450E+00 1.170E-05

END
```

Engineering Systems Development Page 1 of 2
Job: Ex6 15 Aug 1999
DYNAMIC ANALYSIS EXAMPLE 10:19 PM
3D FRAME - 3 bays x 2 bays x 4 storeys

INPUT/ANALYSIS REPORT

```
Job:    Ex6

Title:  DYNAMIC ANALYSIS EXAMPLE
        3D FRAME - 3 bays x 2 bays x 4 storeys
Type:   Space frame
Date:   15 Aug 1999
Time:   10:19 PM

Nodes ............................     60
Members ..........................    116
Spring supports ..................      0
Sections .........................      4
Materials ........................      2
Primary load cases ...............      0
Combination load cases ...........      0

 NODE TABLE NOT PRINTED

 MEMBER TABLE NOT PRINTED

 SECTION PROPERTY TABLE NOT PRINTED

 MATERIAL TABLE NOT PRINTED
```

NODE MASSES

Node	Mass tonne	Node	Mass tonne	Node	Mass tonne	Node	Mass tonne
31	10.000	32	10.000	33	10.000	34	10.000
41	10.000	42	10.000	43	10.000	44	10.000
51	10.000	52	10.000	53	10.000	54	10.000
61	10.000	62	10.000	63	10.000	64	10.000
71	10.000	72	10.000	73	10.000	74	10.000
81	10.000	82	10.000	83	10.000	84	10.000
91	10.000	92	10.000	93	10.000	94	10.000
101	10.000	102	10.000	103	10.000	104	10.000
111	10.000	112	10.000	113	10.000	114	10.000
121	10.000	122	10.000	123	10.000	124	10.000
131	10.000	132	10.000	133	10.000	134	10.000
141	10.000	142	10.000	143	10.000	144	10.000

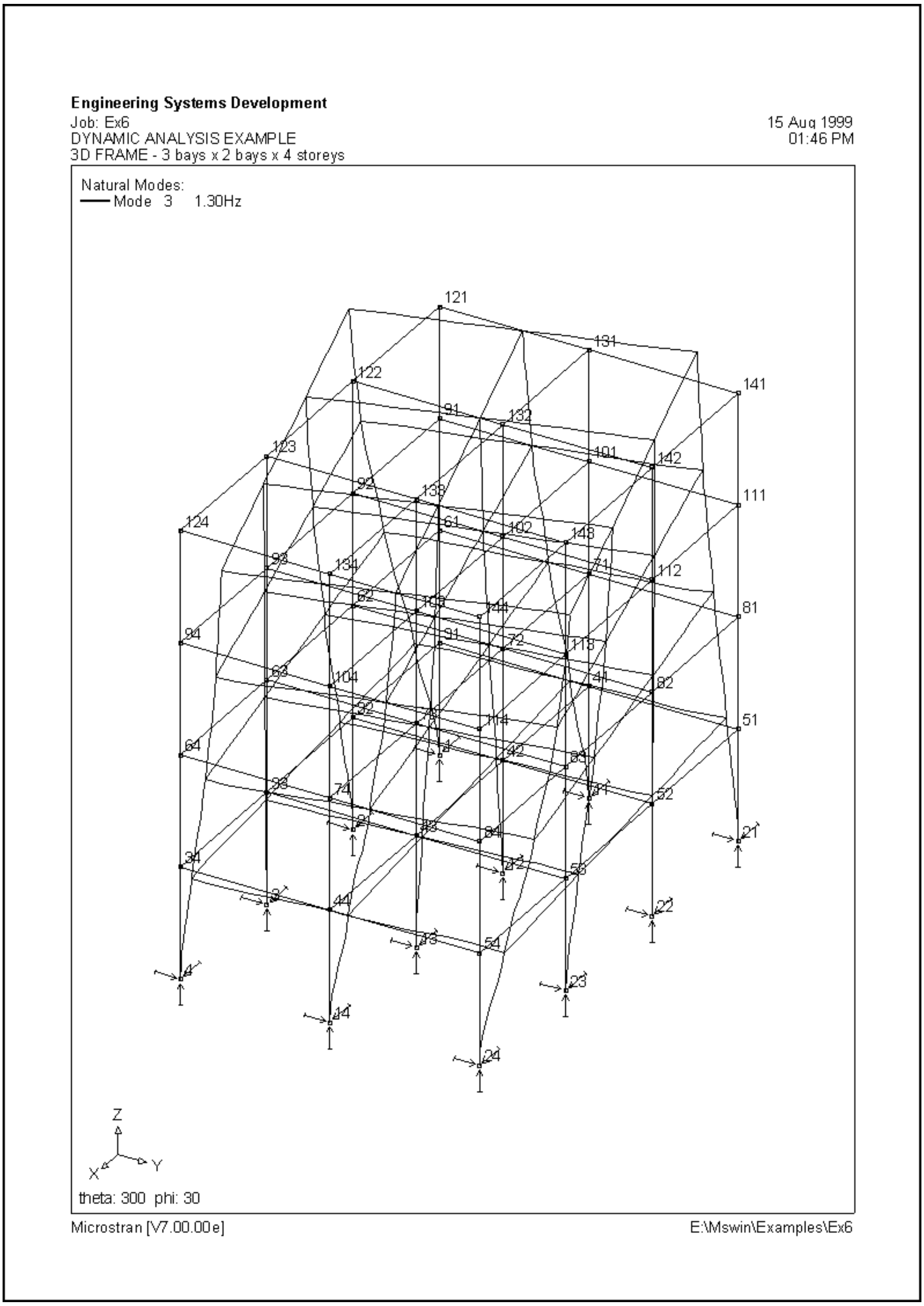
EIGENVECTOR COMPONENTS

Mode: 1 Frequency: 1.24 Hz

Node	X	Y	Z
1	0.0000000	0.0000000	0.0000000
2	0.0000000	0.0000000	0.0000000
3	0.0000000	0.0000000	0.0000000
4	0.0000000	0.0000000	0.0000000
11	0.0000000	0.0000000	0.0000000
12	0.0000000	0.0000000	0.0000000
13	0.0000000	0.0000000	0.0000000
14	0.0000000	0.0000000	0.0000000
21	0.0000000	0.0000000	0.0000000
22	0.0000000	0.0000000	0.0000000
23	0.0000000	0.0000000	0.0000000
24	0.0000000	0.0000000	0.0000000
31	0.0000116	0.2695773	0.0049158
32	0.0000078	0.2711140	0.0049408
33	0.0000010	0.2711140	0.0049408
34	-0.0000027	0.2695773	0.0049159
41	0.0000035	0.2697235	0.0000000
42	0.0000035	0.2712595	0.0000000
43	0.0000035	0.2712595	0.0000000
44	0.0000035	0.2697235	0.0000000

Microstran [V7.00.00e] E:\Mswin\Examples\Ex6.p1

51	-0.0000027	0.2695773	-0.0049159
52	0.0000010	0.2711140	-0.0049408
53	0.0000079	0.2711140	-0.0049408
54	0.0000116	0.2695773	-0.0049158
61	0.0000215	0.5975239	0.0080349
62	0.0000136	0.6008873	0.0080756
63	-0.0000009	0.6008872	0.0080755
64	-0.0000088	0.5975240	0.0080350
71	0.0000039	0.5974408	-0.0000001
72	0.0000039	0.6008008	0.0000000
73	0.0000039	0.6008008	0.0000000
74	0.0000039	0.5974408	0.0000001
81	-0.0000088	0.5975240	-0.0080350
82	-0.0000009	0.6008872	-0.0080755
83	0.0000136	0.6008873	-0.0080756
84	0.0000215	0.5975240	-0.0080349
91	0.0000248	0.8515739	0.0095427
92	0.0000136	0.8563521	0.0095909
93	-0.0000069	0.8563521	0.0095908
94	-0.0000181	0.8515739	0.0095429
101	-0.0000007	0.8514937	-0.0000001
102	-0.0000007	0.8562674	0.0000000
103	-0.0000007	0.8562674	0.0000000
104	-0.0000007	0.8514937	0.0000001
111	-0.0000181	0.8515739	-0.0095429
112	-0.0000069	0.8563521	-0.0095908
113	0.0000136	0.8563521	-0.0095909
114	0.0000249	0.8515739	-0.0095427
121	0.0000240	0.9944311	0.0099764
122	0.0000112	1.0000000	0.0100267
123	-0.0000123	0.9999999	0.0100267
124	-0.0000250	0.9944312	0.0099766
131	-0.0000057	0.9943005	-0.0000002
132	-0.0000057	0.9998639	0.0000000
133	-0.0000057	0.9998639	0.0000000
134	-0.0000057	0.9943005	0.0000002
141	-0.0000250	0.9944312	-0.0099766
142	-0.0000122	1.0000000	-0.0100267
143	0.0000112	1.0000000	-0.0100267
144	0.0000240	0.9944312	-0.0099764



Engineering Systems Development
Job: Ex6
DYNAMIC ANALYSIS EXAMPLE
3D FRAME - 3 bays x 2 bays x 4 storeys
15 Aug 1999
01:46 PM
Natural Modes:
Mode 3 1.30Hz
theta: 300 phi: 30
Microstran [V7.00.00e]
E:\Mswin\Examples\Ex6

Index

A

B

C

D

E

F

N

O

P

Q

R

S

T

U

V

W

X

Y

Z